# openSUSE 12.2

# Start-Up

July 17, 2012

# Start-Up

Copyright © 2006–2012 Novell, Inc. 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 Novell trademarks, see the Novell Trademark and Service Mark list http://www.novell.com/company/legal/trademarks/tmlist.html. All other third party trademarks are the property of their respective owners. A trademark symbol (®, ™ etc.) denotes a Novell trademark; an asterisk (*) denotes a third party trademark.

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

# Contents

About This Guide ix

Part I Installation and Basics 1

1 Installation Quick Start 3
1.1 Welcome to openSUSE. . . . . 3

2 KDE Quick Start 11
2.1 Getting Started. . . . . 11
2.2 Starting Programs. . . . . 12
2.3 Handling Media. . . . . 14
2.4 Customizing Your Desktop. . . . . 15
2.5 Applying Desktop Effects. . . . . 20
2.6 Using Virtual Desktops. . . . . 21
2.7 Managing Files and Folders. . . . . 22
2.8 Browsing the Internet. . . . . 26
2.9 Managing Passwords. . . . . 28
2.10 E-Mailing and Calendaring. . . . . 29
2.11 Instant Messaging with Kopete. . . . . 31
2.12 Starting LibreOffice. . . . . 32
2.13 Viewing PDF Files and Other Documents. . . . . 32
2.14 Searching for Data. . . . . 33
2.15 Creating a CD or DVD. . . . . 35
2.16 Managing Your Digital Image Collection. . . . . 36
2.17 Managing Your Music Collection. . . . . 36
2.18 Logging Out. . . . . 37
2.19 For More Information. . . . . 38

3 GNOME Quick Start 39
3.1 Getting Started. . . . . 39
3.2 Starting Programs. . . . . 41
3.3 Handling Media. . . . . 42
3.4 Customizing Your Desktop. . . . . 42
3.5 Applying Desktop Effects. . . . . 44
3.6 Using Virtual Desktops. . . . . 45
3.7 Managing Files and Folders. . . . . 46

3.8 Browsing the Internet with Firefox. . . . . 48
3.9 Managing Passwords. . . . . 50
3.10 E-Mailing and Calendaring. . . . . 51
3.11 Instant Messaging with Pidgin. . . . . 51
3.12 Starting LibreOffice. . . . . 52
3.13 Viewing PDF Files and Other Documents. . . . . 52
3.14 Searching for Data. . . . . 53
3.15 Creating a CD or DVD. . . . . 55
3.16 Managing Your Digital Image Collection. . . . . 56
3.17 Managing Your Music and Videos. . . . . 57
3.18 Logging Out. . . . . 58
3.19 For More Information. . . . . 59

4 LibreOffice Quick Start 61
4.1 Compatibility. . . . . 61
4.2 LibreOffice Modules. . . . . 61
4.3 Starting LibreOffice. . . . . 62
4.4 LibreOffice Writer. . . . . 64
4.5 LibreOffice Calc. . . . . 67
4.6 Other LibreOffice Modules. . . . . 69
4.7 For More Information. . . . . 73

Part II Managing and Updating Software 75

5 Installing or Removing Software 77
5.1 Definition of Terms. . . . . 77
5.2 Using the KDE Interface (Qt). . . . . 78
5.3 Using the GNOME Interface (GTK+). . . . . 84
5.4 Managing Software Repositories and Services. . . . . 88
5.5 Keeping the System Up-to-date. . . . . 91

6 YaST Online Update 97
6.1 The Online Update Dialog. . . . . 97
6.2 Installing Patches. . . . . 100
6.3 Automatic Online Update. . . . . 101

7 Installing Packages From the Internet 103
7.1 1-Click Install. . . . . 103
7.2 YaST Package Search. . . . . 104

8 Installing Add-On Products 107
8.1 Add-Ons. . . . . 107
8.2 Binary Drivers. . . . . 108

9 Managing Software with Command Line Tools 109
9.1 Using Zypper. . . . . 109
9.2 RPM—the Package Manager. . . . . 118

Part III Administration 127

10 Managing Users with YaST 129
10.1 User and Group Administration Dialog. . . . . 129
10.2 Managing User Accounts. . . . . 130
10.3 Additional Options for User Accounts. . . . . 132
10.4 Changing Default Settings for Local Users. . . . . 137
10.5 Assigning Users to Groups. . . . . 137
10.6 Managing Groups. . . . . 138
10.7 Changing the User Authentication Method. . . . . 139

11 Changing Language and Country Settings with YaST 141
11.1 Changing the System Language. . . . . 141
11.2 Changing the Country and Time Settings. . . . . 144

12 YaST in Text Mode 147
12.1 Navigation in Modules. . . . . 148
12.2 Restriction of Key Combinations. . . . . 149
12.3 YaST Command Line Options. . . . . 149

13 Setting Up Hardware Components with YaST 151
13.1 Hardware Information. . . . . 151
13.2 Setting Up Sound Cards. . . . . 151
13.3 Setting Up a Scanner. . . . . 154

14 Printer Operation 157
14.1 The Workflow of the Printing System. . . . . 158
14.2 Methods and Protocols for Connecting Printers. . . . . 158
14.3 Installing the Software. . . . . 159

14.4 Setting Up a Printer. . . . . 159
14.5 Network Printers. . . . . 163
14.6 Printing from the Command Line. . . . . 165
14.7 Special Features in openSUSE. . . . . 166
14.8 Troubleshooting. . . . . 167

15 Installing and Configuring Fonts for the Graphical User Interface 175
15.1 Adding Fonts. . . . . 175

16 Upgrading the System and System Changes 177
16.1 Upgrading the System. . . . . 177
16.2 For More Information. . . . . 181

Part IV The Bash Shell 183

17 Shell Basics 185
17.1 Starting a Shell. . . . . 185
17.2 Entering Commands. . . . . 186
17.3 Working with Files and Directories. . . . . 189
17.4 Becoming Root. . . . . 192
17.5 File Access Permissions. . . . . 194
17.6 Useful Features of the Shell. . . . . 198
17.7 Editing Texts. . . . . 201
17.8 Searching for Files or Contents. . . . . 203
17.9 Viewing Text Files. . . . . 204
17.10 Redirection and Pipes. . . . . 205
17.11 Starting Programs and Handling Processes. . . . . 206
17.12 Important Linux Commands. . . . . 207

18 Bash and Bash Scripts 217
18.1 What is “The Shell”?. . . . . 217
18.2 Writing Shell Scripts. . . . . 222
18.3 Redirecting Command Events. . . . . 223
18.4 Using Aliases. . . . . 224
18.5 Using Variables in Bash. . . . . 224
18.6 Grouping And Combining Commands. . . . . 226
18.7 Working with Common Flow Constructs. . . . . 227
18.8 For More Information. . . . . 228

A Help and Troubleshooting 229
A.1 Help and Documentation. . . . . 229
A.2 Common Problems and Their Solutions. . . . . 233
A.3 Legal Notice. . . . . 260
A.4 GNU Free Documentation License. . . . . 260

# About This Guide

This manual will see you through your initial contact with openSUSE®. Check out the various parts of this manual to learn how to install, use and enjoy your system.

Installation and Setup
: Guides you through the installation process and the basic configuration of your system. Introduces YaST, the central tool for installation and configuration of your system. Learn how to set up or modify key components of your system and how to connect to the internet.

Basics
: This part mainly addresses to users changing to Linux from other operating systems. It introduces basic Linux concepts such as the user concept, the structure of the file system and access permissions for files and directories. An introduction to the shell is also provided, although you hardly ever need the command line to interact with your system nowadays.

Help and 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 problems on your own.

This manual documents milestone 7 of openSUSE 12.2.

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

For an overview of the documentation available for your product and the latest documentation updates, refer to http://www.novell.com/documentation/opensuse114 or to the following section.

## 1 Available Documentation

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

Start-Up (page i)
: Guides you step-by-step through the installation of openSUSE from DVD, or from an ISO image, gives short introductions to the GNOME and KDE desktops including some key applications running on it. Also gives an overview of LibreOffice and its modules for writing texts, working with spreadsheets, or creating graphics and presentations.

Reference (↑Reference)
: Gives you a general understanding of openSUSE and covers advanced system administration tasks. It is intended mainly for system administrators and home

users with basic system administration knowledge. It provides detailed information about advanced deployment scenarios, administration of your system, the interaction of key system components and the set-up of various network and file services openSUSE offers.

Security Guide (↑Security Guide)
: Introduces basic concepts of system security, covering both local and network security aspects. Shows how to make use of the product inherent security software like AppArmor (which lets you specify per program which files the program may read, write, and execute) or the auditing system that reliably collects information about any security-relevant events.

System Analysis and Tuning Guide (↑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.

Virtualization with KVM (↑Virtualization with KVM)
: This manual offers an introduction to setting up and managing virtualization with KVM (Kernel-based Virtual Machine) on openSUSE. Also shows how to manage VM Guests with `libvirt` and QEMU.

Find HTML versions of most product manuals in your installed system under `/usr/share/doc/manual` or in the help centers of your desktop. Find the latest documentation updates at `http://www.suse.com/documentation` where you can download PDF or HTML versions of the manuals for your product.

# 2 Feedback

Several feedback channels are available:

Bugs and Enhancement Requests
: To report bugs for a product component, or to submit enhancement requests, please use `https://bugzilla.novell.com/`. For documentation bugs, submit a bug report for the component Documentation of the respective product.

  If you are new to Bugzilla, you might find the following articles helpful:

  - `http://en.opensuse.org/Submitting_Bug_Reports`
  - `http://en.opensuse.org/Bug_Reporting_FAQ`

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.de`. 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 typographical conventions are used in this manual:

- `/etc/passwd`: directory names and filenames
- *`placeholder`*: replace *`placeholder`* with the actual value
- `PATH`: the environment variable PATH
- `ls`, `--help`: commands, options, and parameters
- `user`: users or groups
- [], [] + [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
- Dancing Penguins (Chapter Penguins, ↑Another Manual): This is a reference to a chapter in another manual.

# 4 About the Making of This Manual

This book is written in Novdoc, a subset of DocBook (see `http://www.docbook.org`). The XML source files were validated by `xmllint`, processed by `xsltproc`, and converted into XSL-FO using a customized version of Norman Walsh's stylesheets. The final PDF is formatted through XEP from RenderX. The open source tools and the environment used to build this manual are available in the package `daps` that is shipped with openSUSE. The project's home page can be found at `http://daps.sf.net/`.

# 5 Source Code

The source code of openSUSE is publicly available. Refer to `http://en.opensuse.org/Source_code` for download links and more information.

# 6 Acknowledgments

With a lot of voluntary commitment, the developers of Linux cooperate on a global scale to promote the development of Linux. We thank them for their efforts—this distribution

would not exist without them. Furthermore, we thank Frank Zappa and Pawar. Special thanks, of course, goes to Linus Torvalds.

Have a lot of fun!

Your SUSE Team

# Part I. Installation and Basics

1 Installation Quick Start

Use the following procedures to install a new version of openSUSE® 12.2. This document gives a quick overview on how to run through a default installation of openSUSE.

# 1.1 Welcome to openSUSE

For more detailed installation instructions and deployment strategies, see the openSUSE documentation at http://www.novell.com/documentation/opensuse114/.

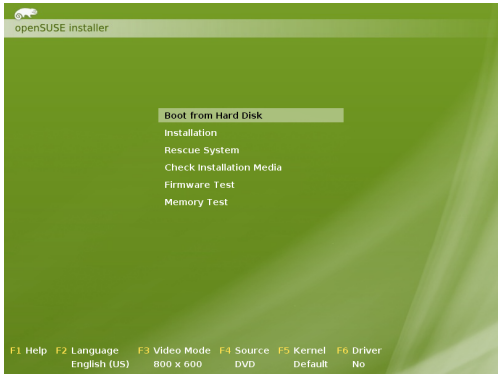
## 1.1.1 System Requirements

- Pentium* III 500 MHz or higher processor (Pentium 4 2.4 GHz or higher or any AMD64 or Intel* EM64T processor recommended)
- 512 MB physical RAM (1 GB recommended)
- 3 GB available disk space (more recommended)
- 800 x 600 display resolution (1024 x 768 or higher recommended)

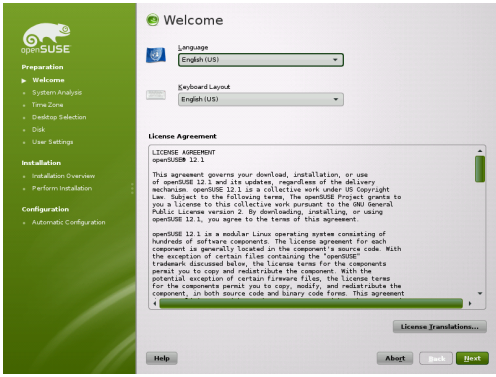
## 1.1.2 Installing openSUSE

Use these instructions if there is no existing Linux system on your machine, or if you want to replace an existing Linux system.

1. Insert the openSUSE DVD into the drive, then reboot the computer to start the installation program.

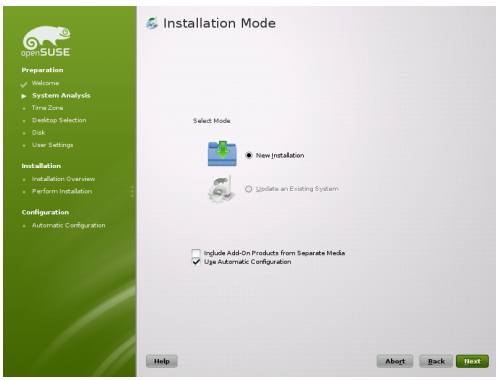


2 Select Installation on the boot screen and press [Enter]. This loads the openSUSE installation program and starts the installation in normal mode.

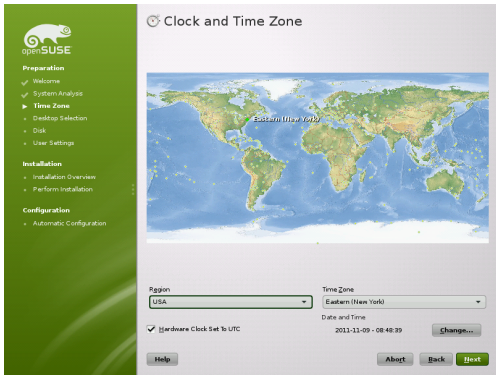



3 Select the language and the keyboard layout to be used during the installation and for the installed system.

Read the License Agreement and proceed with Next if you agree. If you do not agree, press Abort to terminate the installation.

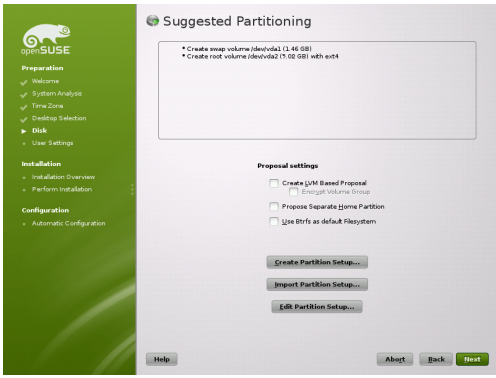



4 Select New Installation and proceed with Next.

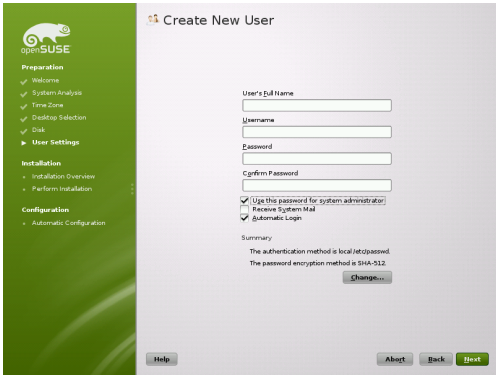



5 Either use the map or the drop down lists for Region and Time Zone to select the clock and time zone to use in your system. Proceed with Next.

6 Select the desktop system you would like to use. KDE and GNOME, are powerful graphical desktop environments similar to Windows, alternative selections are available when checking Other. Proceed with Next.




7 Define a partition setup for openSUSE in this step. In most cases a reasonable scheme that can be accepted without change is proposed. Accept the proposal with Next and proceed with the installation. Experienced users can also customize the proposal (Edit Partition Setup) or apply their own partitioning scheme (Create Partition Setup).

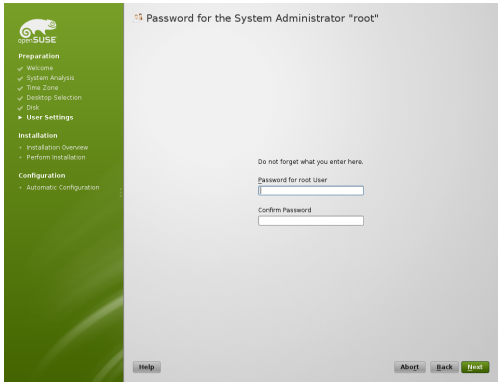



8 Type your first and last name in the User's Full Name field, your login name in the Username field, and your password in the Password field.

For security reasons, your password should be at least eight characters long and should contain uppercase and lowercase letters and numbers. Passwords are case-sensitive. Proceed with Next.

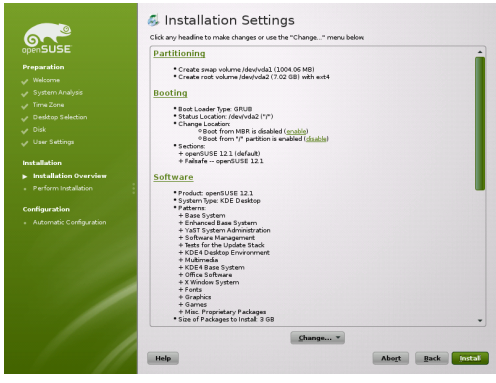
By default, the password you enter here is also used for the system administrator, `root`. Uncheck the relevant option if you prefer to specify a different password for `root` in the next step.

With automatic login enabled, the system boots straight to your desktop with no authentication at all. If you store sensitive data on your system, you should not enable this option as long as the computer can also be accessed by others.



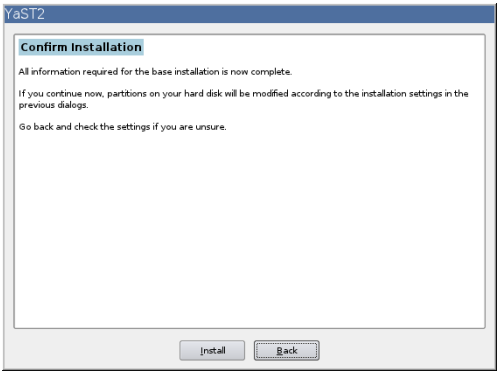

9 Type a password for the system administrator account (called the `root` user). This step is skipped in case you had activated Use this Password for System Administrator in the previous step.

You should never forget the `root` password. After you enter it here, the password cannot be retrieved. It can only be reset with administrative assistance. Proceed with Next.



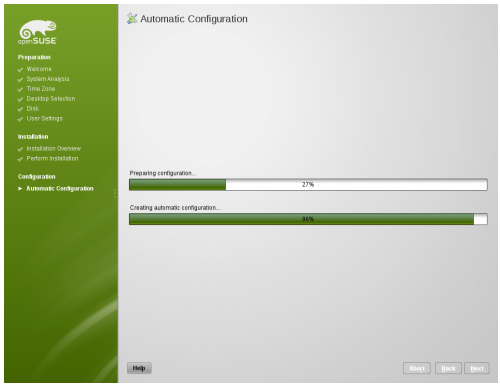

10 Use the Installation Settings screen to review your choices as well as some automatically proposed settings. If necessary, change the settings.

Click Install to proceed. Depending on your software selection you may have to accept additional license agreements by clicking Accept. Before the actual installation starts you will see the following screen:




11 Click Install to start the installation.

12 The installation usually takes between 15 and 30 minutes, depending on the system performance and the selected software scope. During the installation you may either view the product presentation, the release notes containing last-minute information, or the detailed installation log.




13 After completing the installation of all selected software packages and the basic system setup, the openSUSE installation boots into the new Linux system, to perform the automatic system configuration. This routine attempts to configure your network and Internet access and sets up your hardware. This process does not need any interaction.

14 If you have chosen the automatic login function, the desktop directly starts. Otherwise you will see the login screen. Click on your username, and type your password in the Password field. Press [Enter] to login.

openSUSE® provides the tools that Linux* users require in their daily activities. It comes with an easy-to-use graphical environment, the KDE* desktop, that communicates with the underlying Linux system to access and manage files, folders, and programs. openSUSE provides an integrated suite of applications for a wide range of purposes (office, multimedia, Internet). The LibreOffice suite is also included, which allows you to edit and save files in a number of formats. Because the office suite is available for several operating systems, you can use the same data across different computing platforms.

## 2.1 Getting Started

When you start your system, you are usually prompted to enter your username and password. If you did not install the system yourself, check with the system administrator for your username and password.

After logging in to KDE for the first time, you see the KDE desktop, which shows the following basic elements:

Desktop Folder: By default, the desktop folder shows the contents of its `~/Desktop` directory. Whenever you insert a file into this directory, it appears in this view too.

Desktop Context Menu: Right-click an empty area on the desktop to access the context menu for configuring the appearance of the desktop, adding panels or widgets to

the desktop, configuring desktop settings, locking the widgets in their current position, leaving the current session, or for locking the screen.

Desktop Toolbox: Move your mouse pointer to the icon in the upper right corner of the desktop to access the menu that lets you add or lock widgets.

KDE Panel: By default, the panel of your KDE desktop consists of the Main Menu icon on the left, further widgets and program icons, and the panel settings icon on the right. If you hover the mouse pointer over an icon in the panel, a short description is displayed.

Main Menu Click on the left side of the bottom panel to open the main menu. The Main Menu consists of the following elements: a search function at the top and several tabs at the bottom, providing quick access to the key functions of the menu. Additionally, the menu displays your login name and the hostname of your computer.

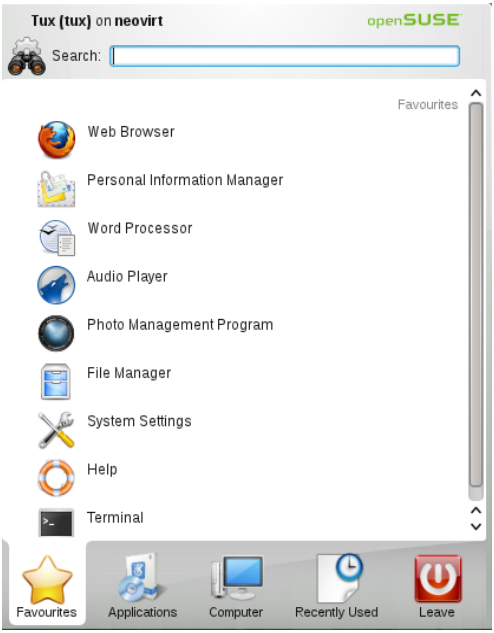
## 2.2 Starting Programs

Start programs from the Main Menu or from the command line, using a shell or the Run Command. Additionally, you can start programs from the desktop or the panel by left-clicking the program icon once.

### 2.2.1 Using the Main Menu

To open the Main Menu, click the Main Menu icon in the panel or press [] + [F1].

Favorites shows a default selection of key programs for quick access, whereas Applications shows all applications installed on your system. To navigate through the menu structure, click an entry and use the arrow icons at the right or the left to switch back and forth. Computer and Recently Used provide quick access to some frequently used places, applications, or documents. Leave shows several options for leaving the session such as logging out, locking the screen (access can only be regained with a password), shutting down or restarting the computer. Some computers offer to suspend your session to RAM or disc.

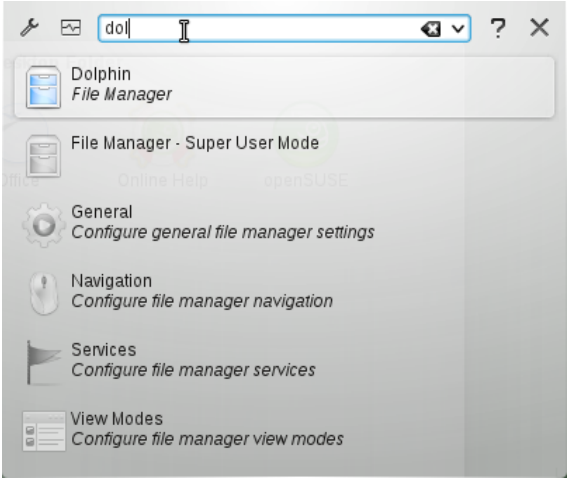



If you often need a program and do not want to dig into the menu structure, you can integrate the icon into the Favorites tab. Right-click the icon on any of the other tabs and choose Add to Favorites.

## 2.2.2 Using the Run Command Dialog

KRunner is a helper application that lets you quickly start programs. Apart from that, it offers a search function for finding applications or locations.

Press [] + [F2] to open the Run Command dialog. Type a command, for example, `dolphin`. While you are typing, the dialog shows a list of applications or actions that match your input. Select one of the results or press [] to start the highlighted application or action. The command to start the application is often (but not always) the application name written in lowercase.




The Run Command dialog allows you also to use different features. Click on the question mark icon to see a list of active features. To configure or get a list of all features, click on the wrench icon. You can use so-called Web shortcuts defined in Konqueror to send search requests directly to a search engine without first opening the browser. Another interesting feature is to calculate a mathematical expression like `=2+2^4-sin(2)`. Very useful is to convert units, for example, from miles per hour into kilometres per hour. Just insert the expression `60 mph in km/h` to get the converted value.

### 2.2.3 Searching for Programs

Both the Main Menu and the Run Command dialog offer a search function that lets you quickly start programs even if you do not already know the exact application name or command. To search for an application, start typing a command or part of the application name in the Main Menu Search field or the input field in the Run Command dialog. Each character you enter narrows down the search.

From the list below the input field, choose the application matching your query.

## 2.3 Handling Media

If you insert a CD or DVD into your drive or plug in a pluggable device (for example, a USB stick or removable hard disk), the device notifier widget opens a small window alerting you to this. It also proposes possible actions, depending on the type of the device.

For example, if you plugged in a digital camera, the actions allow you to download your photos with Gwenview or digiKam. Click the device entry in the widget to access the available actions. To safely remove the device from your computer, use the icon on the right side of the device entry.

## 2.4 Customizing Your Desktop

You can change the way your KDE desktop looks and behaves to suit your personal preferences.

### 2.4.1 Locking and Unlocking Desktop Objects

Desktop elements can be locked in their current position to prevent them from being moved around on the desktop. As long as the desktop elements are locked you cannot add, move, or remove any objects to or from your desktop.

To lock or unlock the desktop elements, click the cashew icon (desktop toolbox) at the upper right corner of the desktop and select Lock Widgets or Unlock Widgets.

Alternatively, right-click an empty patch on the desktop and select Lock Widgets or Unlock Widgets.

### 2.4.2 Changing Individual Desktop Elements

In the following, you can find some examples of how to change individual desktops elements.

Procedure 2.1 Adding Program Icons to the Desktop

To create a link to an application and place it on the desktop or the panel, proceed as follows:

1 Click the Main Menu button and browse to the desired application.

2 Right-click and select Add to Desktop from the context menu that appears. If this menu item is not available, your desktop elements are probably locked. Unlock them first as described in Section 2.4.1, “Locking and Unlocking Desktop Objects” (page 15).

3 To change the icon position on the desktop, left-click the icon and drag it to the desired position.

To delete an icon from your desktop, right-click the program icon and select Remove this Icon.

Procedure 2.2 Adding Widgets to the Desktop or to the Panel

1 To add widgets to you desktop, right-click an empty patch on your desktop and select Add Widgets.

To add a widget to the panel, right-click an empty patch on the panel, and select Panel Options > Add Widgets.

If these menu items are not available, your desktop elements are probably locked. In this case, right-click and select Unlock Widgets first.

2 In the list that appears, select a widget. Use the scrollbar to browse through the available widgets. To limit the selection of widgets, either enter a search term in the input field or choose a Category.

3 To make the selected widget appear on the desktop or in the panel, double-click it and close the dialog box.

4 To remove a widget from your desktop or panel, right-click the widget and select the Remove... entry.

Procedure 2.3 Configuring Desktop Widgets

1 Hover your mouse pointer over the widget until a translucent frame appears around the widget, showing a number of symbols:

If the frame does not appear, your widgets are probably locked. Unlock them first as described in Section 2.4.1, “Locking and Unlocking Desktop Objects” (page 15).

2 To change the widget size, left-click the scale symbol in the frame and keep the mouse button pressed while slowly moving your cursor across the desktop.

3 To rotate the widget in any direction, left-click the arrow symbol in the frame and keep the mouse button pressed while moving your cursor in a circle across the desktop. There is a sticky boundary in the horizontal and vertical positions where you can “clamp” the widget, if you like. Of course, you can also arrange it in any tilted position.

4 To change the contents, settings, or properties of a widget, left-click the wrench symbol in the frame. In the configuration dialog box that appears, set the options according to your wishes.

Procedure 2.4 Adding and Removing Panels

1 If you want multiple panels on the desktop, right-click an empty area on the desktop, select Add Panel and choose a panel type from the list.

If these menu items are not available, your desktop objects are probably locked. In this case, right-click and select Unlock Widgets first.

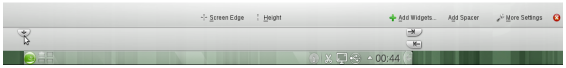
2 Click the cashew icon at the end of the new panel to access the configuration options and customize the panel as described in Procedure 2.5, “Configuring Panels” (page 17).

3 To delete a panel, right-click a blank space in the panel you want to delete and select Panel Options > Remove this Panel. openSUSE requires that at least one panel is left on the desktop.

Procedure 2.5 Configuring Panels

You can customize all KDE panels according to your preferences. Applications and widgets can be added to the quick launch area or the system tray in the main panel or to additional panels. Panel elements and additional panels can be moved to different places or be completely removed at any time.

1 To access any options for the current panel, right-click an empty patch in the panel (or click the cashew icon at the end of the panel) and select Panel Options > Panel Settings. The panel changes view and shows its configurations options.




If the menu item and the icon are not available, your desktop objects are probably locked. To unlock, right-click and select Unlock Widgets first.

2 To move panel objects to a different place within the panel, hover the mouse cursor over the respective panel object (for example, program icon, pager, task manager, system tray). The mouse cursor turns into a crosslines-like

form. Click the object and move the cursor to the desired position in the panel. Click again to fix it at this place.

3 To enlarge the distance between two widgets, use Add Spacer.... A spacer widget appears on the right side. Drag and drop it to the preferred place.

4 To change the panel width, click and drag the small arrow icons at the left and right end of the settings panel. Use the arrow on the left side to set the panel's position and the arrows at the right side to set the maximum and minimum size.

5 To change the panel height, click the Height button and move the mouse cursor to the desired end location.

6 To place the panel at different sides of the screen use the Screen Edge button. Drag and drop the panel to the preferred location.

7 If you need additional widgets on your panel, add them with the Add Widgets... button as described in Procedure 2.2, “Adding Widgets to the Desktop or to the Panel” (page 16).

8 To remove program icons or widgets from the panel, right-click the unwanted object and select the menu item to remove the icon or widget.

The icons in the system tray usually belong to applications running in the background. Therefore you can only remove those icons if you close those applications (right-click the icon and select Quit).

9 To prevent the panel and widgets from being moved accidentally, use Lock Widgets.

10 For additional configurations options like alignment and functions (like auto hide), click More Settings....

11 If all panel changes are according to your wishes, close the panel settings with the red close button on the right side.

Procedure 2.6 Changing the Desktop Background

You can change the desktop background to display images, a slideshow, patterns or colors. It is even possible to use a map of the globe or the weather report as desktop background.

1 Right-click an empty patch of the desktop and select Desktop Settings. A configuration dialog appears.

2 In the View section, choose a type from the Wallpaper drop-down list. The further configuration options depend on the choice of the object to use as background.

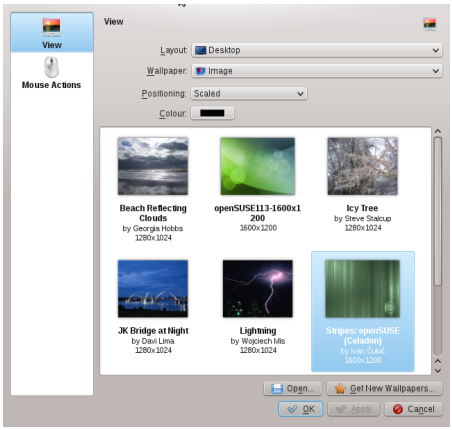
3 For example, to use an image wallpaper:

3a Set Wallpaper to Image.

3b Define the Positioning of the image.

3c To use one of the available wallpapers, either choose a wallpaper from the list or click Get New Wallpapers to download additional wallpapers.

3d To use a custom picture, click Open and select an image file from the file system.



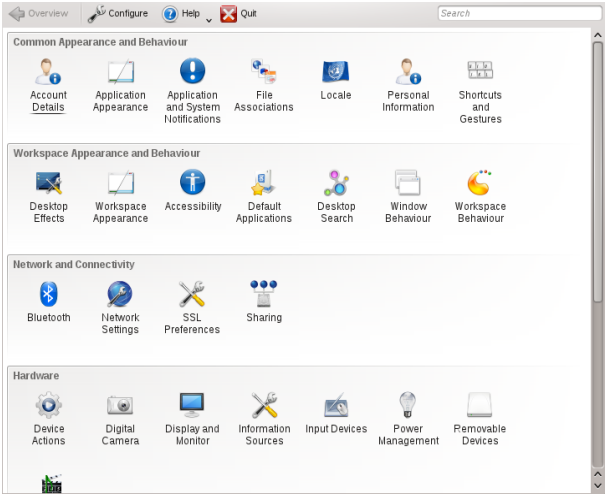

4 Set the other options in the configuration dialog according to your wishes, and click OK to save your changes and leave the configuration dialog.

## 2.4.3 Modifying Personal Settings

Apart from changing individual desktop elements, KDE allows you to personalize your desktop to a very high degree. Possibilities to adjust the overall appearance and behavior of your desktop can be found in the Personal Settings configuration dialog.

Start the Personal Settings from the Main Menu by selecting Favorites > Configure Desktop.

The settings are grouped into different categories. Hover over an icon to see more details. Performing tasks in some areas of the personal settings requires system administrator (`root`) permissions.




Change the settings as desired. No changes take effect until you click Apply. To discard changes in the recent view that you have not yet applied, click Reset. To reset all items in the recent view to the default values, click Defaults.

To get back to the start-up view (showing all categories) again, click Overview at the top left corner of the dialog. You can also enter a search string at the top of the window (for example, Screen Saver) to find the category which holds options related to the search string. Each character you enter in the Search field further narrows the search.

For easier overview of all categories, you can also switch to a tree view by clicking Configure and selecting another View Style.

## 2.5 Applying Desktop Effects

If your computer supports a 3D graphics card with Xgl, you can use graphical effects like turning your desktop into a rotating 3D cube, enabling translucent or transparent windows, and zooming in and out of the desktop screen. You can also use other window effects such as shadows, fading, and transformations. If you do not have a 3D graphics card, some effects may be disabled or reduce the performance of your system. If the performance of your system is too low, disable the effects for your current session with [] + [] + [F12].

To configure effects on your desktop, proceed as follows:

1 Start the System Settings as described in Section 2.4.3, “Modifying Personal Settings” (page 19).

2 Select Desktop Effects from the Workspace Appearance and Behavior category and choose Enable Desktop Effects. If your graphic card does not support 3D, you will be notified.

3 Optionally, change some options below Common Settings.

4 On the All Effects tab, select the effects you want to activate. Click the information icon for a short explanation of the respective effect. If an effect is activated, the wrench icon on the right gives access to further configuration options like assigning keyboard shortcuts to the effect.

5 Use Apply to enable the effects permanently.

## 2.6 Using Virtual Desktops

The desktop environment allows you to organize your programs and tasks on several virtual desktops. If you often run a lot of programs simultaneously, this minimizes the number of windows to arrange on your screen. You might, for example, use one desktop for e-mailing and calendaring and another for word processing and graphics applications.

Procedure 2.7 Moving an Application to Another Virtual Desktop

You can display a running application on one or all virtual desktops, or move it to other desktops.

1 Open the application.

2 Right-click the title bar of the application.

3 Click To Desktop.

4 Select the desktop on which to place the application.

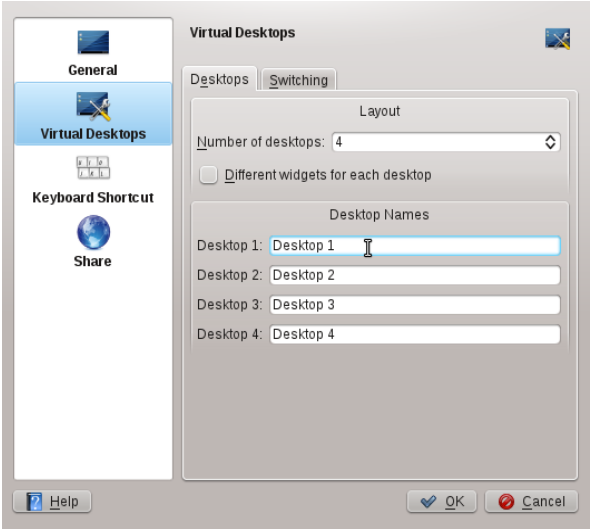
5 To switch to a certain desktop, click the desired desktop in the pager in the panel. Switch to the next or previous desktop by placing the mouse cursor on an empty patch on the desktop and move the mouse wheel.

Procedure 2.8 Configuring Virtual Desktops

You can increase the number of virtual desktops, change the default names, assign activities, configure effect animations or assign keyboard shortcuts for switching virtual desktops.

1 To add additional desktops, right-click the pager in the panel and select Add Virtual Desktop. A new desktop is shown in the pager.

2 For any configuration options, right-click the pager and select Pager Settings.




3 Use the options on the General screen to configure the appearance of the pager in the panel.

4 Desktop names, keyboard shortcuts for switching desktops and the switching type can be configured on the Virtual Desktops screen.

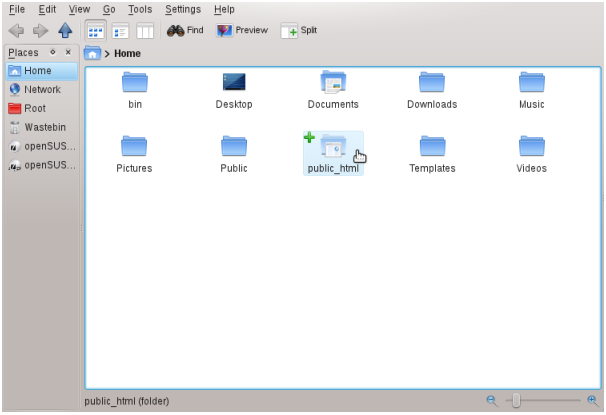
5 Set the options according to your wishes and click OK to apply the changes and to close the configuration dialog.

# 2.7 Managing Files and Folders

With KDE 4, Dolphin has replaced Konqueror as the default file manager. Konqueror remains a Web browser (refer to Section 2.8, “Browsing the Internet”(page 26). Start Dolphin, from the Main Menu by selecting Favorites > File Manager or press [] + [F2] and enter `dolphin`.

## 2.7.1 Dolphin Main Window

The Dolphin main window consists of the following elements:




Menu Bar: The menu bar holds menu items for actions like copying, moving, or deleting files, changing views, starting additional tools, defining your settings, and getting help.

Toolbar: The toolbar provides quick access to frequently used functions that can also be accessed via the menu. If you hover the mouse pointer over an icon, a short description is displayed.

Location Bar: The location bar displays the path to the current directory. It is available in two versions: one shows the path to the current directory with icons for every parent folder in a “bread crumb” view. Click any icon in the bread crumb view to change to that directory. The second version of the location bar shows the path to the current directory as a string of text you can edit.

Panels: By default, Dolphin shows the Places panel. It allows quick access to some often-used places like your home directory, the root directory of the file system (/), the trash bin, and removable media. There are several other panels you can add to the main window by clicking View > Panels.

Display Field (Working Space): The display field shows the contents of the selected directory or file. By default, Dolphin displays the contents of your home directory on start-up. By default, clicking a folder or file in Dolphin directly starts an action: Dolphin loads the file into an application for further processing or opens the folder.

Status Bar: Shows the file type and size of the currently selected object and allows you to enlarge or diminish the size of the icons or entries shown in the display field.

## 2.7.2 Managing Files and Folders with Dolphin

To perform actions like copying, moving, creating or deleting files, you need appropriate permissions to the folders and files involved in your action.

To copy, move, or delete a file or folder, proceed as follows:

1 In order to select one or multiple files and folders in Dolphin, move your mouse pointer over the file or folder but do not click. A green cross appears on the upper-left edge. If you click it, the object is selected. To deselect again, click the red minus icon that appears if the object is currently selected.

   Alternatively, press [] and click the objects you want to select or deselect.

2 Right-click and select Copy or Cut from the context menu.

3 Navigate to the destination folder in which to insert the object.

4 To create a new folder at the current location, select File > Create New > Folder or press [F10]. Enter a folder name in the new window and press [].

5 To insert the object you copied or cut in Step 2 (page 24), right-click the destination folder in the main display field and select Paste. The object is copied or moved there.

6 To delete a file or folder, right-click the object in the main display field and select Move to Trash from the context menu. The object is moved to the trash bin. From there, you can restore it if necessary or delete the object permanently.

To quickly filter for certain filenames in the current directory, press [] + [I] to add the Filter input field to the bottom of the Dolphin main window. Type any part of the filename you are searching for to see all files in the current directory containing the search string. For more detailed and advanced searches, press [] + [F] to use the desktop search. For more information, refer to Section 2.14.2, “Using Strigi Desktop Search” (page 34).

## 2.7.3 Configuring Dolphin

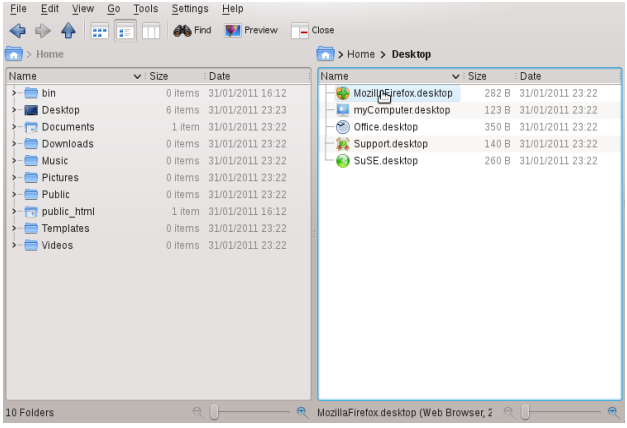
Dolphin offers many options for adjusting the view and the overall settings according to your preferences.

Procedure 2.9 Changing the View

1 To switch from the bread crumb view to the editable version of the location bar, press [F6]. Enter a path to a directory by typing it in. After typing an address, press [] or click on the background of the bread crumb view.

   To delete the contents of the location bar click the black X symbol on the left. To switch back to bread crumb view, press [] + [L].

2 To change the view of the currently displayed folder, either click Icons, Details, or Columns in the toolbar. Dolphin remembers the selected view for each folder. Click Split or press [F3] to view the contents of the current folder in two separate columns. Now you can navigate to a different directory in each column and easily drag or drop objects or compare the contents of directories. Press [F3] again to leave the split view.




3 To make Dolphin also show hidden files, select View > Show Hidden Files.

4 To view more details about the files (like access permissions or ownership) in the working space, select View > Additional Information and enable the corresponding options.

5 To add further directories to the Places panel, drag a folder from the working space to the Places panel and drop it there. Right-click and use the context menu to hide, edit or remove entries from Places.

6 To add more panels to the main window, select View > Panels and select additional panels such as Folders, or Terminal.

Detach the panels from the main Dolphin window by clicking the left icon at the top of each panel. Click the panel's title bar and drag it to another place on the desktop. To reintegrate the panel into the Dolphin window, click the left symbol at the top of the panel again.

Procedure 2.10 Changing Dolphin's Overall Behavior

If you want to change Dolphin's overall behavior or view, select Settings > Configure Dolphin and explore the options offered in the Dolphin configuration dialog.

1 To use the same view mode for all folders, click General in the left sidebar and activate Use Common View Properties for All Folders. Click View Modes in the left sidebar and adjust the options for the individual view modes. Click Apply to save the changes.

2 If you want Dolphin to show a different directory on start-up, or if you want to permanently use the editable location bar instead of the bread crumb view, click Startup in the left sidebar and change the relevant options.

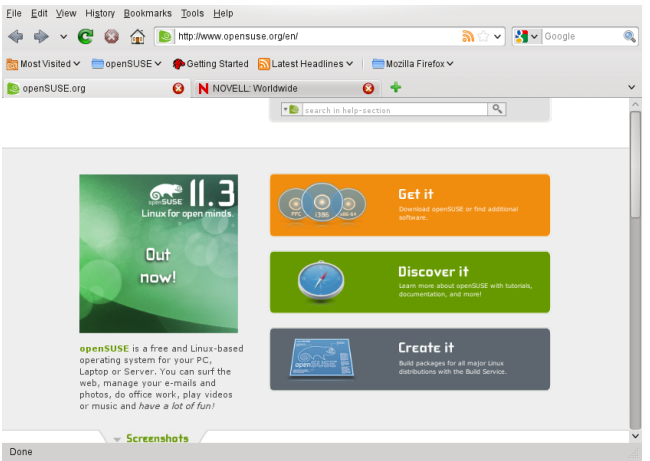
3 To configure which entries and services to show in the Dolphin context menu, click General. On the Context Menu tab, activate the entries to show. Click Services in the left sidebar and activate or deactivate the respective services to appear in the context menu.

4 Click OK to save the changes and to close the Dolphin configuration dialog.

## 2.8 Browsing the Internet

Apart from Konqueror (KDE's default Web browser), openSUSE also includes the default Firefox Web browser. To start Konqueror or Firefox, press [] + [F2] and enter `konqueror` or `firefox`.

With features like tabbed browsing, pop-up window blocking, and download and image management, both browsers combine the latest Web technologies. Their easy access to different search engines helps you find the information you need.




Enter a URL in the location bar to start browsing. To open a new, empty tab, press [] + [T]. To open a link in a new tab, click the link with your middle mouse button. Right-click the tab itself to access more tab options. You can create a new tab, reload one or all existing tabs, or close them. You can also change the sequence of the tabs by dragging and dropping them to the desired position.

### 2.8.1 Finding Information

Both browsers offer different kinds of search options: you can find information on the Web or you can search the current Web page for keywords.

Procedure 2.11 Searching the Web

1. To start a search on the Web, click the left icon in the search bar (located next to the location bar) to open a list of search engines.
2. Select the desired engine and type your search keywords into the input field.
3. Press [] to start the search.
4. To search the current Web page with Firefox or Konqueror, press [] + [F] to open a find bar at the bottom of the window. Enter your search keyword there and use the buttons beneath the bar to search in different directions or to configure search options such as Match Case.

Konqueror offers predefined Web shortcuts for quickly searching the Web. For example, to search the Web for `yast` using the Google search engine, just type `gg: yast` into the Konqueror location bar (or the Run Command dialog) and press [].

You can also define individual Web shortcuts in Konqueror and Firefox as described in Section 2.8.3, “Configuring Preferences” (page 27).

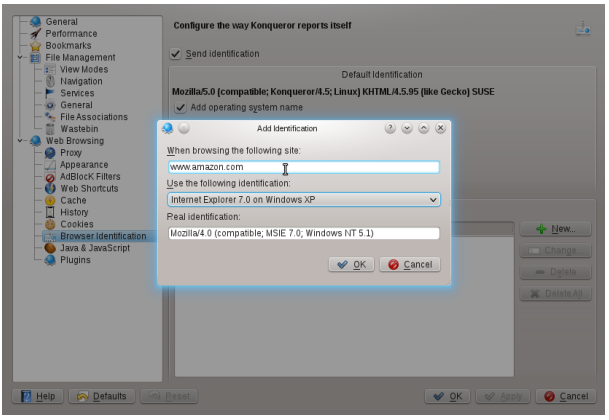
## 2.8.2 Downloading Files from the Web

If you download a file with Firefox or Konqueror, a dialog window appears asking you if you want to save the file or to open it with a specific application. Also choose if to remember the action you selected for this type of file. By default, Firefox saves the file to the folder configured in the Firefox Preferences and shows your finished downloads in the Downloads dialog.

To open the downloaded files directly from there, right-click and select Open. To clean up the history of downloaded files, click Clear List.

## 2.8.3 Configuring Preferences

Adjust preferences in Konqueror or Firefox. In Konqueror, access the configuration dialog with Settings > Configure Konqueror. To change the behavior of Konqueror as Web browser, select one of the Web Browsing sub categories in the left sidebar to show the options for the relevant category. For example, you can “hide” Konqueror's real identity and make Konqueror identify as a different browser (for example, Internet Explorer*) for certain Web sites you visit. To do so, select Browser Identification, and click New to add a new, site-specific identification.




You might also want to configure the Web shortcuts you can use with Konqueror (and the Run Command dialog). In the Konqueror configuration dialog, click Web Shortcuts to see which shortcuts are already defined. Click New to define new shortcuts. Leave the Konqueror configuration dialog with OK to apply your changes.

In Firefox, select Edit > Preferences to open the Firefox Preferences. Click the icons in the upper display field to access the options for the relevant category. For example, change the default download folder on the General page, or adjust the pop-up blocking feature on the Content page. Click OK to apply the changes.

To configure a Web shortcut for a search in Firefox, browse to the Web page with the desired search field. Right-click the search field and choose Add a Keyword for this Search.... Enter a Keyword and leave the dialog with OK.

## 2.9 Managing Passwords

When you enter a password in a KDE application for the first time (in KMail or Konqueror, for example), you are asked if you want to store the password. If you click Yes, KWallet wizard starts by default. KWallet is a password management tool that can collect all passwords and store them in one or more encrypted files (called `wallets`).

To activate KWallet, select Basic Setup and click Next. Select Yes, I wish to use the KDE wallet to store my personal information and enter a password. This is your master password for the default wallet (named `kdewallet`) that is created during the next step.

---

WARNING: Protect Your KWallet Password

If you forget your master password for any wallet, it cannot be recovered. Furthermore, anyone who knows your master password can obtain all information contained in the respective wallet.

---

Click Finish to close the wizard. If you started the KWallet wizard initially from another application like Konqueror or KMail, KWallet will prompt you to enter the password once again in order to store the password for this application in the newly created wallet. After initial configuration you can open your wallet at any time to view, search, delete, or create entries. Normally you do not need to insert an entry manually. KDE recognizes if a resource requires authentication and KWallet starts automatically, prompting you for the KWallet password. However, you can add additional data if you wish. To configure any KWallet options, press [] + [F2] and enter `kwalletmanager`. In the KDE Wallet Manager that opens select Settings > Configure Wallet.

Whereas KWallet is designed to centrally manage passwords for several KDE applications, Firefox also offers the ability to store data when you enter a username and a password on a Web site. If you accept by clicking Remember, the password will be stored on your hard disk in an encrypted format. The next time you access this site, Firefox will automatically populate the login fields.

To review or manage your passwords in Firefox, click Edit > Preferences > Security > Saved Passwords....

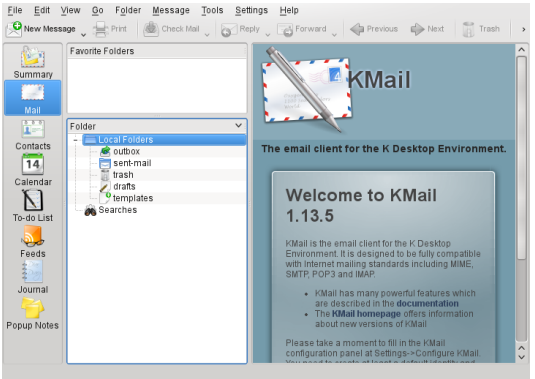
## 2.10 E-Mailing and Calendaring

For reading and managing your mails and appointments, you can use Kontact as your personal information management tool (PIM). Kontact combines KDE applications like

KMail, KOrganizer, and KAddressBook into a single interface. This gives you easy access to your e-mails, calendar, address book, and other PIM functionality. Kontact can also manage multiple e-mail accounts.

## 2.10.1 Starting Kontact for the First Time

To start Kontact, press [] + [F2] and enter `kontact`. To access one of its components like KMail, click the corresponding icon in the left sidebar.




Before you can send or receive mails, you must configure an e-mail identity and account:

1. From the menu, select Settings > Configure Kontact....

2. In the left side pane under Mail, click Identities and select the identity that KMail has automatically created for you. Modify it by entering your full name and e-mail address and confirming your changes.

3. Click Accounts in the left side pane under Mail and Add at least one account on the tabs for Receiving and Sending mails. (Optionally, you can configure multiple mail accounts).

   Usually, the following information is required for both incoming and outgoing mail: Account Type, Server Name and Port. Depending on the account, you may also need to set encryption or authentication options. If you are not sure about the settings or items to select, consult your Internet Service Provider or system administrator.

4. Confirm the setup and leave the configuration dialog with OK.

To write a new mail, click the Mail icon in the Kontact main window and press [] + [N] to open the mail composer. After you have finished your mail, click Send to send

it immediately. In case you have configured multiple e-mail accounts, keep the Send button pressed, then select the account from which you want to send the mail.

## 2.11 Instant Messaging with Kopete

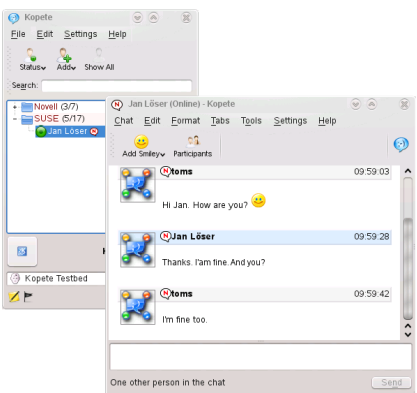
Kopete is an online messenger application that allows multiple partners connected to the Internet to chat with each other. Kopete currently supports a number of common messenger protocols, such as AOL* Instant Messenger (AIM), Gadu-Gadu, GroupWise® Messenger, ICQ, Jabber*, SMS, and Yahoo!*. To be able to use instant messaging (IM), you must register with a provider offering IM services and configure a Kopete account.

To start Kopete, press [] + [F2] and enter `kopete`. If started for the first time, click the wrench icon at the bottom of the Kopete window to create an account. Kopete guides you through the setup steps like selecting the messaging service, and entering the respective account information like user name or ID, server name and port and any authentication details like passwords. If you have not registered with the provider for the selected messaging service yet, click Register New Account. In the browser window that opens, enter your user data to register. Switch back to Kopete and enter the data received on registration. Complete the configuration of your account by clicking Finish.

If you decided to go online after configuring your account, you can now add contacts in the main Kopete window. If the Kopete window shows `offline` , first click File > Set Status > Online and enter your password if you are prompted for it. After connecting, click File > Add Contact and select the Kopete account for which you want to add contacts. Enter the contact data or search for a contact and click OK.

To view all your contacts, even if they are offline at the moment, select Settings > Show Offline Users.

To start a chat with someone, click the desired contact and type your message in the lower part of the chat window. Press [] to send the message. The upper part of the window displays the messages you have sent and received.



## 2.12 Starting LibreOffice

The office suite LibreOffice offers a complete set of office tools, including word processor, spreadsheet, presentation, vector drawing, and database components. Because LibreOffice is available for a number of operating systems, you can use the same data across different computing platforms.

To start LibreOffice press [] + [F2] and enter `libreoffice`. To create a new document, select File > New and choose the type of document to create. To open an existing document, select Open and choose the appropriate file from the file system.

Find more information in the LibreOffice Quick Start.

## 2.13 Viewing PDF Files and Other Documents

Documents that need to be shared or printed across multiple platforms can be saved as PDF (Portable Document Format) files, for example, in the LibreOffice suite. View them with Okular, the default KDE document viewer.

### 2.13.1 Using Okular Document Viewer

Apart from PDF files, Okular allows you to view a great number of file formats, such as PostScript, several image formats, OpenDocument format (ODF), OpenDocument text (ODT), some eBook formats (ePub), and even Microsoft* Compiled HTML Help (CHM). Okular also provides support for bookmarks, annotations, form fields and multimedia content, and the rotation of pages.

Start Okular from the Main Menu or press [] + [F2] and enter `okular`.

To open a document, select File > Open and choose the desired file from the file system. Navigate through the document by using the navigation icons at the top or bottom of the window. Depending on which icon you click on the navigation panel on the left, the sidebar either shows a table of Contents, a Thumbnail view of each page, the Reviews, or your Bookmarks for this file. To filter for any text listed in the sidebar or the overall document, enter a string into the input field at the top of the sidebar. If you want to select and copy text or images from the file in Okular, click the Selection icon in the toolbar and select one of the options from the context menu. Click the Browse icon to switch back to browsing the document.

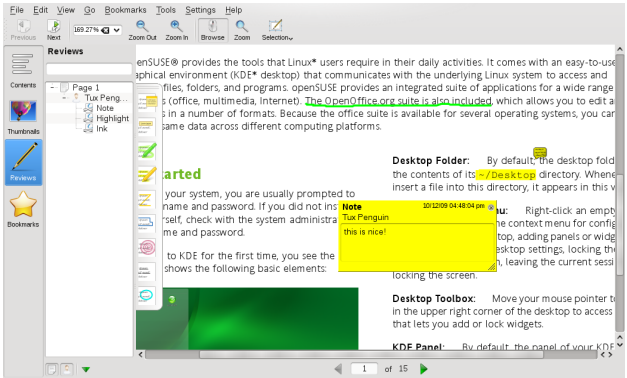
Procedure 2.12 Working with Bookmarks and Annotations

With Okular, you can review a document by highlighting certain text parts or adding annotations or bookmarks that Okular then attaches to the file in the form of metadata. Note that the annotations and markers you add are not stored within the document, so you cannot print them or pass them on to other users.

1. To add a bookmark for a page, click the page in the sidebar or in the main display field and press [] + [B]. The page is added to the Bookmarks list on the

left sidebar. Right-click the bookmark entry to access a context menu for going to this bookmark or renaming, or removing it.

2 To create an annotation for a page, press [F6] and select one of the annotation tools from the toolbar that appears. The annotation is added to the list of Reviews and is flagged with the login name of the user who created it. Use the icons at the bottom of the sidebar to group the annotations by page, by author or to show the annotations for the current page only.




3 To open a pop-up note and add text for an annotation (or to remove an annotation), right-click the annotation entry in the Reviews list and select the corresponding menu item. The annotations or markers you added are automatically attached to the file, you do not need to save them.

### 2.13.2 Using Acrobat Reader

If Acrobat Reader is not installed by default, install the `acroread` package with YaST. To start Acrobat reader, press [] + [F2] and enter `acroread`. Click File > Open, locate the desired PDF file and click Open to view the file.

## 2.14 Searching for Data

KDE provides more than one application for finding data on your computer or in the file system. One application is KFind, which is described below.

### 2.14.1 Using KFind

To perform basic and advanced searches, use KFind. Start it from the Main Menu by clicking Applications > Find Files/Folders or press [] + [F2] and enter `kfind`.

Procedure 2.13 Searching by Name

To use a filename (or a part of it) as the search keyword, proceed as follows:

1 Enter the filename or a part of the filename in the Named input field. Use wild cards, such as an asterisk (*), to indicate missing characters in the filename.

2 Specify the folder in which to search for the file. Either enter a path to the folder in Look in or click Browse to locate the folder.

3 To search subfolders, check Include Subfolders.

4 Click Find to start the search. The results are displayed in the lower part of the dialog. To directly open the result, just click it. Right-click it to choose one of the options from the context menu.

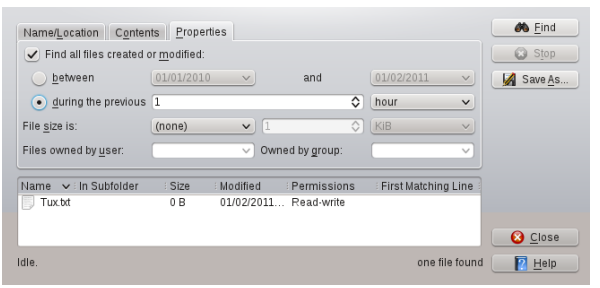
Procedure 2.14 Searching by Contents or Properties

For a more detailed search, you can also specify further options, such as a text the file must contain or the creation or modification date of a file. To search for content or properties, proceed as follows:

1 On the Name/Location tab, specify at least the path to search for the file. You can leave Named empty.

2 To search for certain content in a file, click the Contents tab. In Containing Text, enter the word or phrase the file to find must contain. This option does not work for all types of files.

3 To search for properties such as creation or modification date of the file, click the Properties tab and choose the options you want.




4 Click Find to start the search.

## 2.14.2 Using Strigi Desktop Search

Strigi Desktop Search offers advanced search capabilities by additionally searching for metadata such as tags, ratings, and even source URLs for downloads. It requires the services `nepomuk` and `strigi` to run. To activate these services, press [] + [F2] and enter `strigi`. Choose Nepomuk/Strigi server Configuration to start the configu-

ration dialog. Enable the two services on the tab Basic Settings and leave the dialog by clicking Apply and OK.

Once activated, you can use KRunner ([] + [F2]) to search for files by just typing a keyword. A more advanced search interface is available in the Dolphin file manager. Press [] + [F] in Dolphin to activate the desktop search. You can filter the search results for Filename, Content or location.

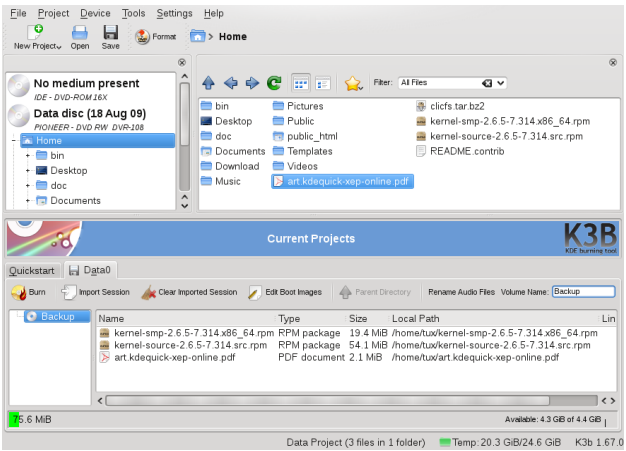
## 2.15 Creating a CD or DVD

If you possess a CD or DVD writer, you can burn files to a CD or DVD with K3b. To create a data CD or DVD, proceed as follows:

1 Press [] + [F2] and enter `k3b`. K3b opens.

2 Select File > New Project > New Data Project from the main menu.

3 Use the tree view in the top left part of the window to search for the files or folders to burn. When they appear in the top right part of the window, drag and drop them into the Current Projects window.




4 Insert a writable CD or DVD into the device then click the Burn icon.

5 Check the settings on the Writing tab. Usually the defaults should work well for almost all files. If you wish, you can change more options (for example the Volume Name) on the Filesystem tab.

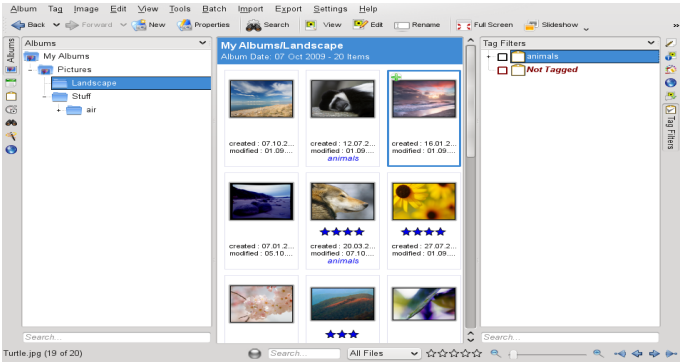
6 Click Burn.

## 2.16 Managing Your Digital Image Collection

With digiKam, it is easy to manage your digital images: download your images from the camera, edit and improve them, organize them in albums (or flag them with tags for easy retrieval, independent of folders or albums), archive them on CD or DVD, or export them to a Web image gallery.

digiKam also contains a set of useful batch functions and plug-ins, allowing you to convert multiple images to various different formats, rename multiple images, or automatically improve them with functions such as red eye reduction, speck removal, and hot pixel removal. Various filter and effects plug-ins help create works of art from your digital images.

To start digiKam, press [] + [F2] and enter `digikam`. On initial start-up, the first-run wizard appears to guide you through the setup.

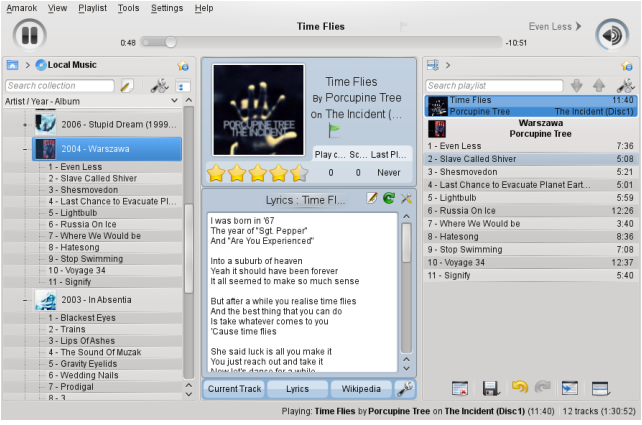



digiKam's main window contains a tree view on the left and shows thumbnails of the images on the right. For quick access to your images, use the sidebar on the left, which allows you to switch between different views, for example, sorting the images according to Albums, Dates, Tags or Searches. To enter digiKam's viewing and editing mode, double-click an image thumbnail.

## 2.17 Managing Your Music Collection

KDE's Amarok music player allows you to play various audio formats, create playlists, listen to streaming audio broadcasts of radio stations on the Internet and subscribe to podcasts. The file types supported depend on the engine used for Amarok.

To start Amarok, press [] + [F2] and enter `amarok`. On first start, Amarok asks for the folders where to look for your music files.




Amarok's main window shows a sidebar on the left providing different views: your local music collection, some addresses to listen to podcasts or streams from the Internet, your playlists and a file browser. The middle part, the context browser, displays information about the current track, the Wikipedia page of the artist and the song's lyrics by default. The right part of the window shows the current playlist. To play music, just drag and drop items from any of the sidebar browsers to the playlist area. Double-click an item in the playlist to start playback.

If your music files are properly tagged (containing at least information about the artist and album), you can make use of several nice Amarok features. To automatically retrieve the album cover images from Amazon, select Tools > Cover Manager and click Fetch Missing Covers. The next time you play a track from this album, the cover is displayed in the context browser and on the on-screen display. To learn more about the artist, click on the Wikipedia button in the middle area to make Amarok search for the appropriate Wikipedia article.

To configure the use of another engine (or various other Amarok features), select Settings > Configure Amarok.

## 2.18 Logging Out

Log out (and leave the system running), restart or shut down the computer.

### 2.18.1 Logging Out or Switching Users

Click the Main Menu button on your panel and select Leave > Logout. This ends your session but leaves the system running. To start a parallel session as a different user, select Leave > Switch User instead.

### 2.18.2 Restarting or Shutting Down the Computer

Click Leave, then select one of the following options:

Sleep
: Only available if your computer provides power management functionality. Pauses your computer without logging you out. All your data and the session data is saved to RAM.

Hibernate
: Only available if your computer provides power management functionality. Pauses your computer without logging you out. All your data and the session data is saved to disk before the system is laid to rest. It is thus protected against data loss should you lose power in the meantime.

Restart
: Restarts your computer.

Shutdown
: Logs you out of the current session, then turns off the computer.

## 2.19 For More Information

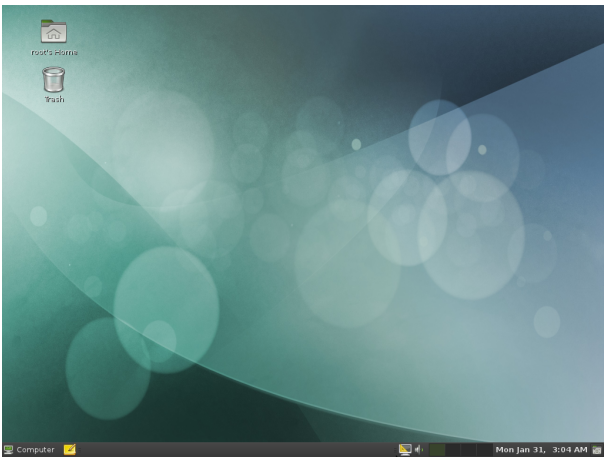
To learn more about KDE and KDE applications, also refer to http://www.kde.org/ and http://www.kde-apps.org/.

openSUSE® provides the tools that Linux* users require in their daily activities. It comes with an easy-to-use graphical environment, the GNOME* desktop, that communicates with the underlying Linux system, to let you access and manage files, folders, and programs. openSUSE provides an integrated suite of applications for a wide range of office, multimedia, and Internet activities. The LibreOffice suite, which allows you to edit and save files in a number of formats, is also included. The office suite is available for several operating systems. Therefore, you can use the same data across different computing platforms.

# 3.1 Getting Started

When you start your system, you are usually prompted to enter your username and password. If someone else installed your system, check with your system administrator for your username and password.

After logging in to GNOME for the first time, you see the GNOME desktop, offering the following basic elements:




Desktop Icons: Access programs and features on your system by double-clicking an icon. Right-click an icon to get additional menus and options. By default, the desktop features several key icons, including your personal Home folder and a trash can for deleted items. Other icons representing devices on your computer, such as CD drives or USB flash disks, may also be present on the desktop, and you can add as many

icons on your desktop as you like. If you double-click your Home folder, the Nautilus file manager starts and displays the contents of your home directory.

Bottom Panel: The desktop includes a panel at the bottom of the screen. This panel contains the Computer menu, the Tomboy Notes applet, a taskbar to display buttons for all currently running applications, and the system tray. You can also add applications and applets to the panel for easy access.

Main Menu: Click Computer on the left side of the bottom panel to open the main menu. Commonly used applications appear in the main menu along with recently used applications. Click the Documents button to display your recent documents, or click the Places button to display your favorite places (such as your home directory, your desktop, or available network servers). Click More Applications to access additional applications, listed in categories. Use the options on the right to access help, open the GNOME Control Center, run YaST, install additional software, lock your screen, log out of the desktop, shut down the system, or check the status of your hard drive and network connections.

System Tray: The right side of the bottom panel shows some smaller icons, including the system clock that displays the current date and time, the Workspace Switcher, the volume control, and icons for several other helper applications.

Taskbar: By default, all applications running on the current desktop are displayed in the taskbar (the area in the middle of the panel between the Computer button and the system tray). You can access these applications by clicking their name in the taskbar. Right-click the name of the application to see options for moving, restoring, or minimizing the window.

Desktop Menu: Right-click an empty spot on the desktop to display a menu with various options. Click Create Folder to create a new folder. Use Create Launcher to create a launcher icon for an application. Provide the name of the application and the command for starting it, then select an icon to represent it. You can also change the desktop background, open a terminal, create a document, and align desktop icons.

### 3.1.1 Modifying Desktop Panels

The bottom panel can be customized to meet your individual needs, and additional panels can be added and configured to further personalize your desktop.

To add a new panel, right-click a blank space in the bottom panel and select New Panel. To delete a panel, right-click a blank space in the panel and select Delete This Panel. openSUSE requires that at least one panel is left on the desktop.

### 3.1.2 Adding and Removing Panel Icons

1. Right-click a blank space on the panel and select Add to Panel.
2. Browse to the desired program and double-click it to add it to the panel.
3. To remove a program icon from a panel, right-click its icon in the panel and select Remove From Panel.

## 3.2 Starting Programs

Start programs from the Main Menu or from the command line, using a shell or the Run Command. Additionally, you can start programs from the desktop or the panel by left-clicking the program icon once.

### 3.2.1 Using the Main Menu

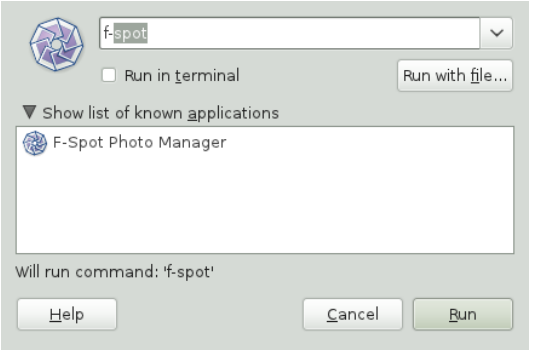
To run a program in openSUSE, click Computer on the panel to open the main menu screen. If the program you want does not appear in the main menu screen, click More Applications to view a list of all available applications grouped in categories. You can also limit the list to show only applications with a specific name by entering all or part of the name into the Filter field. Click an entry in the list to start the corresponding program.

You can also add an icon to your panel that opens a more traditional version of the main menu. Right-click a blank space on the panel, select Add to Panel, then double-click Traditional Main Menu.

If you already know the name of an application but are not sure how to start it from the main menu, use the Search field in the main menu. Click Computer, type a portion of the application name in the Search field, then press []. If the application is installed on your system, the name of the application appears in the Desktop Search dialog box. Click the name to start the program.

### 3.2.2 Using the Run Application Dialog

Press [] + [F2] to open the Run Application dialog. Type a command, for example, `f-spot`, and press [] or click Launch to start the application. The command to start the application is often (but not always) the application name written in lowercase.




## 3.3 Handling Media

If you insert a CD or DVD into your drive or plug in a pluggable device (for example, a USB stick or removable hard disk), GNOME opens Nautilus and shows the content of the media.

## 3.4 Customizing Your Desktop

Easily add, delete, and create shortcut icons on your desktop. You can also change icon properties and the desktop background to suit your needs.

### 3.4.1 Adding Program Icons to Your Desktop

1 Click Computer.

2 Browse to the desired program.

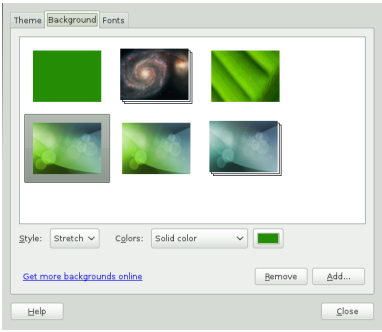
3 Click and drag the icon to the desktop and position it as desired.

To delete an icon from your desktop, simply click the program icon and press the [Delete] key on your keyboard.

### 3.4.2 Changing the Desktop Background

1 Right-click the background.

2 Select Change Desktop Background.




3 To select a wallpaper from the pre-installed images, browse to and select the desired image. After you select it, the background changes automatically.

4 To add an image, select Add, browse to the desired image, then click Open.

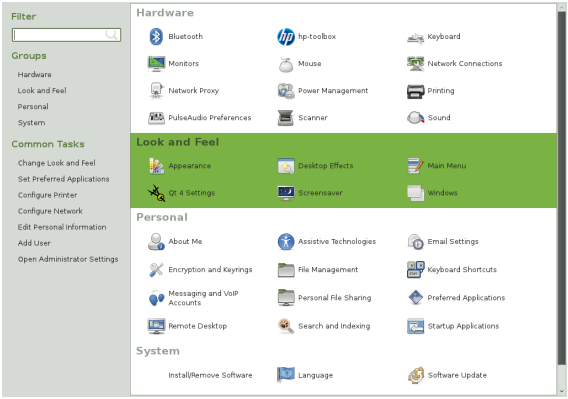
5 To display a colored background, select the solid color rectangle from the top of the list; then, from Colors, select the fill pattern and the desired colors.

6 Click Close.

### 3.4.3 Using the GNOME Control Center

In addition to letting you change individual desktop elements, GNOME lets you extensively personalize your desktop. You can find more settings to adjust the overall appearance and behavior of your desktop in the GNOME Control Center. There, you can also change fonts, keyboard and mouse configurations, regional and language settings, parameters for your Internet and network usage, and more.

To start the Control Center, click Computer, then click Control Center on the right of the main menu.




## 3.5 Applying Desktop Effects

If your computer supports a 3D graphics card with Xgl, you can use graphical effects like turning your desktop into a rotating 3D cube, enabling translucent or transparent windows, and zooming in and out of the desktop screen. You can also use other window effects such as shadows, fading, and transformations. If you do not have a 3D graphics card, some effects may be disabled or reduce the performance of your system. If the performance of your system is too low, disable the compositing effect for your current session with [] + [] + [F12].

To configure the effects on your desktop, proceed as follows:

1 Start the personal settings window as described in Section 3.4.3, “Using the GNOME Control Center” (page 43).

2 Select Look and Feel > Desktop Effects and choose Enable Desktop Effects. If your graphic card does not support 3D, you will be notified.

3 Select the effects in one of the tabs.

## 3.6 Using Virtual Desktops

The desktop environment allows you to organize your programs and tasks on several virtual desktops. If you often run a lot of programs simultaneously, this minimizes the number of windows to arrange on your screen. You might, for example, use one desktop for e-mailing and calendaring and another for word processing and graphics applications.

Procedure 3.1 Moving an Application to Another Virtual Desktop

You can display a running application on one or all virtual desktops, or move it to other desktops.

1 Open the application.

2 Right-click the title bar of the application.

3 Click Move to Another Workspace.

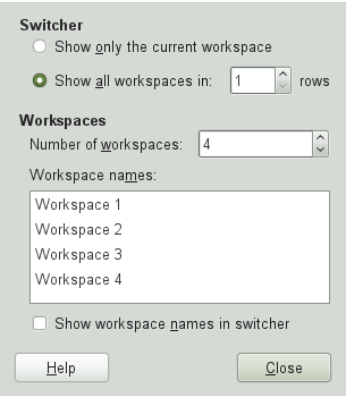
4 Select the desktop on which to place the application.

5 To switch between desktops, click the desired desktop in the pager in the panel.

Procedure 3.2 Adding Additional Virtual Desktops

Some users might need more desktops than are provided by default. To add additional desktops :

1 Right-click the pager in the panel and select Preferences. A configuration dialog appears:

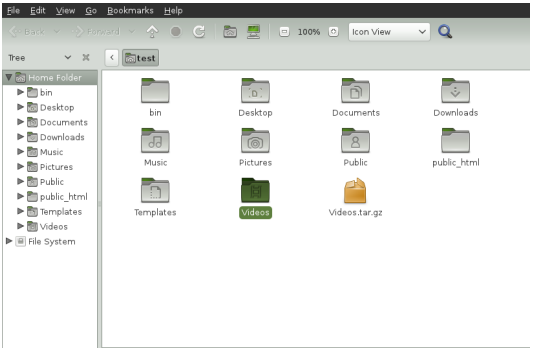



2 Decrease or increase the Number of Workspaces.

3 If you want to change the names of the desktops double click on the list entries and enter the new names.

## 3.7 Managing Files and Folders

Use the Nautilus File Manager to create and view folders and documents, run scripts, and create CDs of your data. Double-click your home directory icon on the desktop to open Nautilus and see the contents of your home directory.




The elements of the Nautilus window include the following:

Menu Lets you perform most tasks.

Toolbar  Lets you quickly navigate among files and folders, and provides access to files and folders.

Location Bar  Lets you locate files, folders, and URI sites. The button on the left side toggles between button-based and text-based display.

Side Pane  Lets you navigate or display information about the selected file or folder. Use the drop-down list to customize what is shown in the pane. The list includes ways to view information about files, perform actions on files, attach emblems to files, view a history of recently visited sites, and display your files in the tree system.

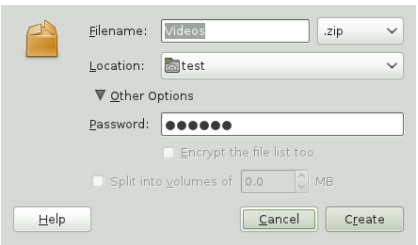
View Pane  Displays folders and files. Use the options on the View menu to increase or decrease the size of content in the view pane and to display items as a list or as icons.

Status Bar  Displays the number of items in a folder and gives the available free space. When a file is selected, the status bar displays the filename and size.

## 3.7.1 Archiving Folders

If you have files you have not used recently but want to keep on your computer, you can compress the files into a tape archive (TAR) format.

1 In the Nautilus view pane, right-click the folder you want to archive and select Compress.




2 Accept the default archive filename or provide a new name.

3 Select a file extension from the drop-down list. Use `tar.gz` for the most common archive form.

4 Specify a location for the archive file, then click Create.

To extract an archived file, right-click the file and select Extract Here.

## 3.7.2 Using Bookmarks

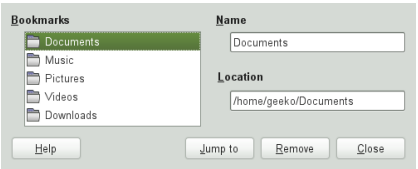
Use the Nautilus bookmarks feature to mark your favorite folders.

1 Open the folder you want to create a bookmark for.

2 Click Bookmarks > Add Bookmark. The bookmark is added to the list, with the folder name as the bookmark name.

3 To select an item from your bookmarks list, click Bookmarks, then click the desired bookmark in the list.

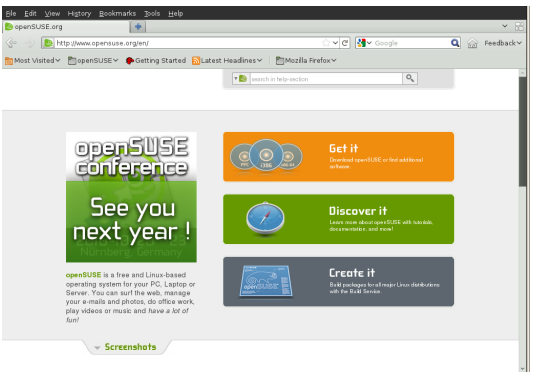
You can also organize your bookmarks list by clicking Bookmarks > Edit Bookmarks and making your selections in the dialog box.




To change the order of your bookmarks, click a bookmark shown in the Edit Bookmarks dialog and drag it to the desired location.

## 3.8 Browsing the Internet with Firefox

Firefox* is becoming the Web browser of choice. It has all the familiarity of other browsers, plus added features such as security and privacy tools.




To start Firefox, click Computer > Firefox Web Browser.

With features like tabbed browsing, pop-up window blocking, and download and image management, Firefox combines the latest Web technologies. Its easy access to

different search engines helps you find the information you need. Enter a URL in the location bar to start browsing.

To open a new empty tab in Firefox, press [] + [T] or the “+” button next to the last tab . Type a new URL to browse. To open a link in a new tab, click the link with your middle mouse button. Right-click a tab to access more tab options. You can create a new tab, reload one or all existing tabs, or close a single tab or all tabs. You can also change the sequence of the tabs by dragging and dropping them to a new position.

## 3.8.1 Finding Information

1 To start a search on the Web type your search keywords in the integrated search box on the right of the location bar, then press []. The results are displayed in the active window or tab.

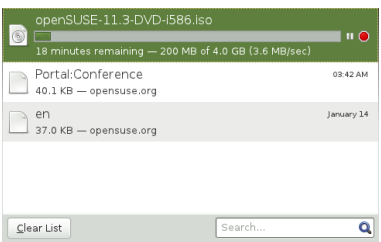
2 To use a different search engine than the default, click the icon in the search box to open a list of other search engines.

3 Click the desired engine, then press [] to start the search.

To search the current Web page for words, press [] + [F] to open the Find bar at the bottom of the window. Enter your search keyword there and use the buttons to the right of the box to search in different directions or to select all hits in the text.

## 3.8.2 Downloading Files from the Web

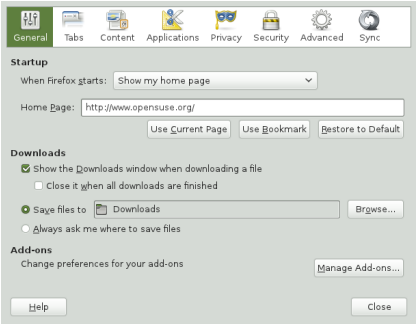
If you download a file with Firefox, the Firefox download manager starts and saves this file to the folder configured in Firefox Preferences. By default, your desktop is the destination folder for downloaded files. Firefox shows your finished downloads in the Downloads window.




Open the downloaded files directly from the Downloads window or from the destination folder. To clean up the history of downloaded files, click Clean Up.

## 3.8.3 Configuring Preferences

To adjust the default download folder or to activate or modify the pop-up blocking feature, click Edit > Preferences.




Here also configure many other settings such as appearance, language, privacy, and tab options. Click the icons and set the options on each page according to your preferences. Click Close to apply the changes.

## 3.9 Managing Passwords

GNOME allows you to store your passwords and encryption keys in a keyring. This is useful, if you want to store passwords for different Web sites.

To create a new keyring, proceed as follows:

1. Press [] + [F2] and enter `seahorse`.
2. Select File > New.
3. Choose Password Keyring and proceed with Continue.
4. Enter a name for your keyring. A dialog box opens.
5. Enter your password, confirm, and click OK.

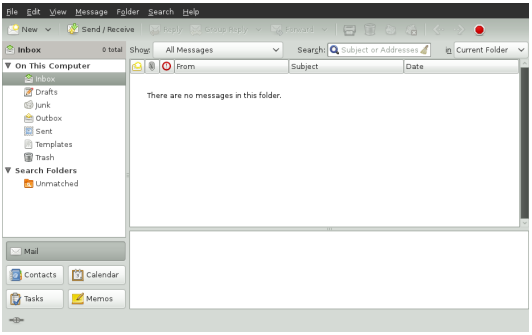
You can create as many keyrings as you like. For example, you can create one for private and one for business. To store passwords in your keyring, proceed as follows:

1. Press [] + [F2] and enter `seahorse`.
2. Select File > New.
3. Choose Store Password and proceed with Continue.
4. Select your keyring, enter description and password.
5. Finish with Add.

# 3.10 E-Mailing and Calendaring

For reading and managing your mail and events, openSUSE offers you Evolution™, a groupware program that makes it easy to store, organize, and retrieve your personal information.

Evolution seamlessly combines e-mail, a calendar, an address book, and a task list in one easy-to-use application. With its extensive support for communications and data interchange standards, Evolution can work with existing corporate networks and applications, including Microsoft* Exchange.




To start Evolution, click Computer > Evolution Mail and Calendar.

The first time you start it, Evolution prompts you with a few questions as it sets up a mail account and helps you import mail from your old mail client. It shows you how many new messages you have and lists upcoming appointments and tasks, as well as the current weather and news from news feeds. The calendar, address book, and mail tools are available in the shortcut bar on the left.

Press [] + [N] to open a new item for whatever part of Evolution you are working in. In mail, this creates a new message. If you are in the address book, [] + [N] creates a new contact card, and in the calendar, it creates a new appointment.

For more information on using Evolution, click Help > Contents in any Evolution window.

# 3.11 Instant Messaging with Pidgin

Pidgin is a multiprotocol instant messaging (IM) client. It is compatible with AOL* Instant Messenger (AIM), Google Talk, GroupWise® Messenger, ICQ, IRC, MSN Messenger, Jabber/XMPP, Yahoo!*, and other networks.

With Pidgin, you can log in to multiple accounts on multiple IM networks simultaneously. Pidgin also supports many features of the various networks, such as file transfer, away messages, and typing notification.

To start Pidgin, click Computer > More Applications > Internet > Pidgin, or press [Alt] + [F2] and enter `pidgin`.

To add an account to Pidgin, either use the initial configuration wizard or select Accounts > Manage Accounts from the Buddy List. Click Manage Accounts, click Add, select the protocol you want, and type your login information in the appropriate fields. Activate the account by checking the Enabled checkbox in the account list.

Add contacts by clicking Buddies > Add Buddy. An assistant appears to help with creation. However, you must be online and connected to the selected messaging service to add a contact to your list.

To start a chat, double-click the desired contact and type your message in the lower part of the chatting window. Press [] to send the message. The upper part of the window displays the messages you have sent and received.

## 3.12 Starting LibreOffice

The LibreOffice office suite offers a complete set of office tools, including word processing, spreadsheet, presentation, vector drawing, and database components. Because LibreOffice is available for a number of operating systems, you can use the same data across multiple platforms.

To start LibreOffice, click Computer > OpenOffice.org Writer. To create a new document, click File > New, then choose the type of document you want to create. To open an existing document, click Open, then select the file you want from the file system.

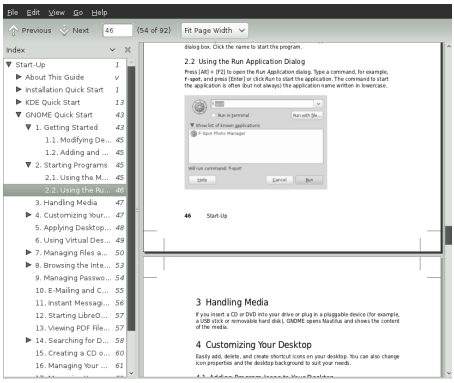
Find more information in the LibreOffice Quick Start.

## 3.13 Viewing PDF Files and Other Documents

Documents that need to be shared or printed across platforms are often distributed as PDF (Portable Document Format) files. PDF files can be created using the LibreOffice suite or other applications. The GNOME PDF viewer is called Evince. It can be used to view PDFs and many other document formats, like PostScript, DjVu, DVI, and multi-page TIFFs.

To start Evince, press [Alt] + [F2] and enter `evince` .

1 To view a PDF file, double-click the PDF file.



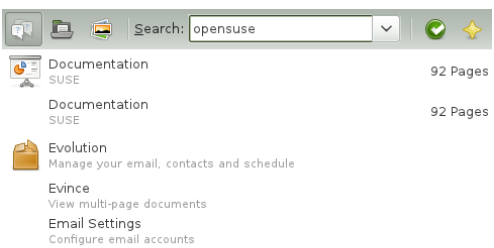

2 Use the navigation icons at the top of the window to navigate through the document. If the PDF document provides bookmarks, you can access them in the left panel of the viewer.

# 3.14 Searching for Data

GNOME provides more than one application for finding data on your computer or in the file system.

## 3.14.1 Using Desktop Search

To locate files or programs on your computer, click Computer. Enter a search term in the Search field and press []. The results are displayed in the Tracker Search Tool.




For a more advanced search tool, use Desktop Search (Beagle Search). To start it, click Computer > More Applications > System > Search.

You can use the results lists to open a file, forward it via e-mail, or display it in the file manager. Simply right-click an item in the results list and select the option you want. The options available for an item depend on the type of file it is. Clicking a file in the list displays a preview of the file and information such as the title, path, and date the file was last modified or accessed.

Use the Find In menu to limit your search to files in a specific location, such as your address book or Web pages, or to display only a specific type of file in your results list. The View menu lets you sort the items in your results list according to name, relevance, or the date the file was last modified.

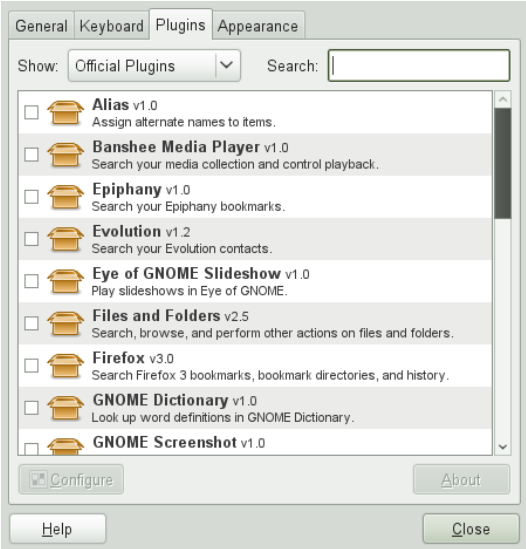
## 3.14.2 Using GNOME Do

GNOME Do describes itself as an “intelligent launcher that makes performing common tasks on your computer simple and efficient.” It is extensible with plug-ins.

Start it by pressing [Super] + [Space]. The [Super] is normally located on the left and right [Windows] or [Penguin] key.

GNOME Do allows you to perform different actions. For example, if you type `b`, it displays a starter button for the Banshee musicplayer:

To customize GNOME Do to your needs, select the arrow button on the upper-right corner of the window and select Preferences. It gives you a list of plug-ins to search for e-mails in Evolution, listen to your music, etc.

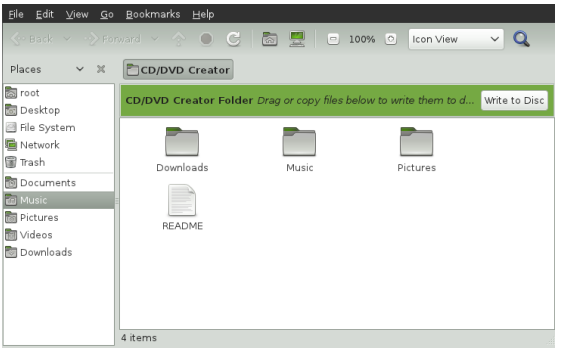



## 3.15 Creating a CD or DVD

If your system has a CD or DVD writer, use the Nautilus file manager to burn CDs and DVDs.

1 Click Computer > More Applications > System > CD/DVD Creator, or just insert a blank disk.

2 Copy the files and folders you want to put on the CD or DVD into the Nautilus CD/DVD Creator window.




3 Click Write to Disk.

4 Modify the relevant information in the Write to Disk dialog box, or accept the default values, then click Write. The files are burned to the disk. This could take a few minutes, depending on the amount of data being burned and the speed of your burner.

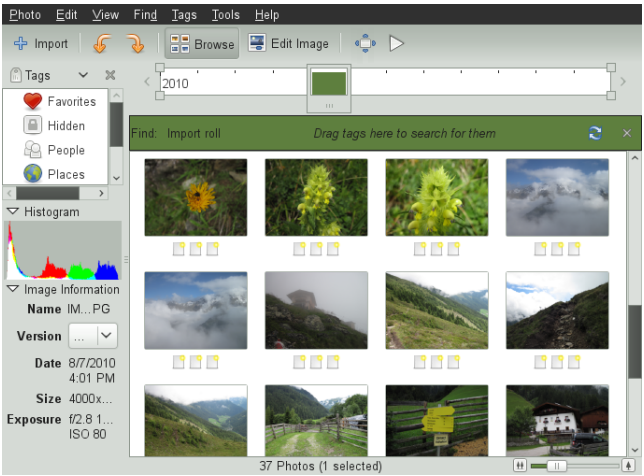
You can also use Brasero or Banshee to burn data and audio CDs.

## 3.16 Managing Your Digital Image Collection

F-Spot is a management tool for your collection of digital images, tailored for the GNOME desktop. It allows you to assign different tags to your images in order to categorize them, and offers various image editing options. For example, you can remove red-eye, crop, and adjust brightness and colors. F-Spot supports all important image formats, including JPEG, PNG, TIFF, and several vendor specific RAW formats.

You can import photos from your hard drive, your digital camera, or your iPod. You can also use F-Spot to create photo CDs, generate a Website gallery, or export your photos to your Flickr, 23, Picasa Web, or SmugMug account.

To access F-Spot, click Computer > F-Spot Photo Browser or press [] + [F2] and enter `f-spot`. The first time you run F-Spot, you must define the location from where you want to import images into your collection. If you already have a collection of images stored on your hard drive, enter the path to the directory and (optionally) include subfolders. F-Spot imports these images into its database.




Thumbnails of your images are displayed in the right part of the window, and detailed information for a selected image is displayed in a sidebar to the left. By default, your photos are displayed in reverse-chronological order, so your newest photos are always at the top. You can sort your photos in chronological order or reverse-alphabetical order by clicking View > Reverse Order.

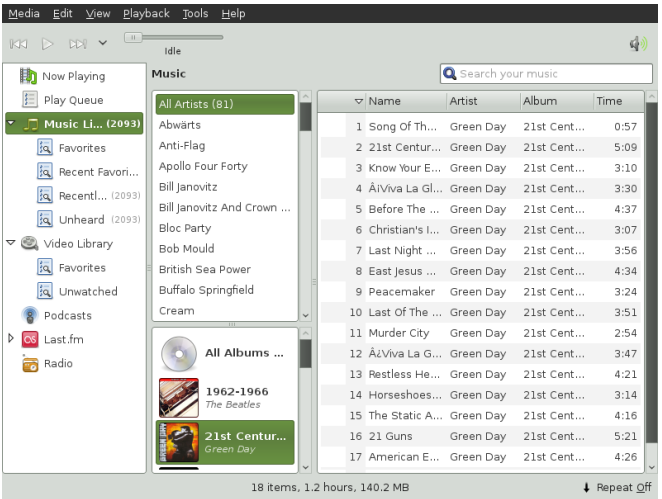
A menu bar at the top of the window allows you to access the main menus. A toolbar below offers several different functions depicted by a matching icon.

## 3.17 Managing Your Music and Videos

Use Banshee to import CDs, sync your media collection to a digital media player, play music directly from a digital media player, create playlists with songs or videos from your library, create audio and data CDs from subsets of your library, and subscribe to, download, and listen to your favorite podcasts. Banshee also has an Internet Radio plug-in that allows you to listen to audio streams.

To open Banshee, click Computer > Banshee Media Player or press [] + [F2] and enter `banshee-1`.

When you start Banshee for the first time, you need to import your media to make them available in the library. To do so, select Media > Import Media from the menu. Choose an import source and click Import Media Source. After successfully importing your media, you can access them from your library.




To play a song or a movie, simply select it in the library and click the play button. You can also use the buttons in the upper left corner to pause a song or play the next or previous song. Use the loudspeaker button on the right to adjust the volume.

Banshee also has an integrated CD player. When you insert a music CD, your CD title appears in the left panel. Select the title and click the Play button to play your full CD.

To create audio and MP3 CDs, select the songs you want, then click the Edit > Write CD from the menu.

# 3.18 Logging Out

When you are finished using the computer, you can log out and leave the system running, or restart or shut down the computer.

## 3.18.1 Logging Out or Switching Users

Click Computer > Logout, then select one of the following options:

Log out  Logs you out of the current session and returns you to the login screen.

Switch User  Suspends your session, allowing another user to log in and use the computer.

## 3.18.2 Restarting or Shutting Down the Computer

Click Computer > Shutdown, then select one of the following options:

Shutdown  Logs you out of the current session, then turns off the computer.

Restart  Logs you out of the current session, then restarts the computer.

Suspend  Puts your computer in a temporary state that conserves power. The state of your session is preserved, however, including all applications you have running and all documents you have open.

Hibernate  Suspends your session, using no power until the computer is restarted. The state of your session is preserved, however, including all applications you have running and all documents you have open.

## 3.19 For More Information

This guide gave you a short introduction to the GNOME desktop and some key applications running on it. To discover more, refer to the other manuals available for openSUSE (at `http://www.suse.com/documentation/opensuse113/` or in your installed system under `/usr/share/doc/manual`).

To learn more about GNOME and GNOME applications, refer to `http://www.gnome.org/`.

The LibreOffice Novell Edition contains improved functionality and additional features not available with the standard version.

## 4.1 Compatibility

LibreOffice can work with documents, spreadsheets, presentations, and databases in many other formats, including Microsoft Office™. They can be seamlessly opened like other files and saved back to the original format. Though some work has already been invested in interoperability, occasionally there are still formatting issues. If you have problems with your documents, consider opening them in the original application and resaving them in an open format such as RTF for text documents. In case of migration problems with spreadsheets however, it is advisable to resave them as Excel files and use this as intermediate format (with CSV format you would lose all cell formatting and CSV sometimes leads to incorrect cell type detection for spreadsheets).

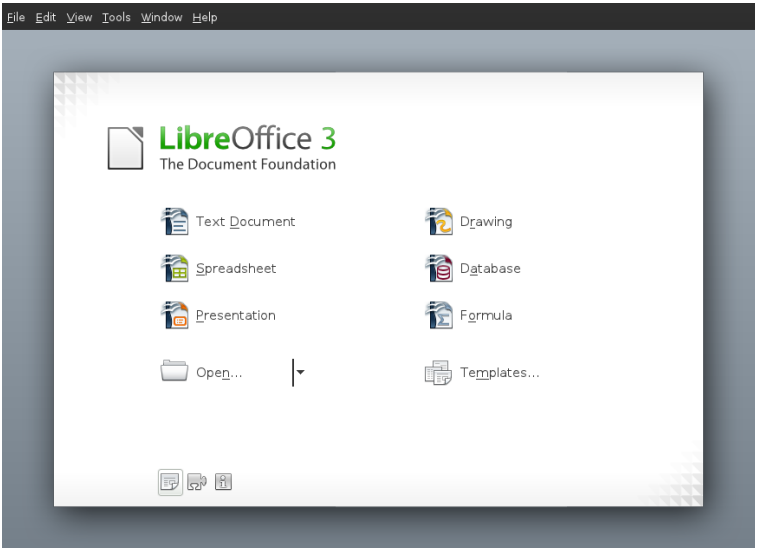
## 4.2 LibreOffice Modules

LibreOffice consists of several application modules (subprograms), which are designed to interact with each other. All modules feature the same graphical user interface and similar functionality.

Table 4.1 The LibreOffice Modules

| Module | Usage |
|---|---|
| Writer | Text document module |
| Calc | Spreadsheet module |
| Impress | Presentation module |
| Base | Database module |
| Draw | Vector graphics module |
| Math | Mathematical equation module |

# 4.3 Starting LibreOffice




To start the LibreOffice suite, press [] + [F2], type `libreoffice` and press [Enter]. In the Welcome dialog, select the module you want to open. If an LibreOffice application is open, you can start any of the other applications by clicking File > New > Name of Application.

## 4.3.1 Saving LibreOffice Files

1. To save a new file, select File > Save or Save as from the Menu Bar or use the shortcut [] + [] + [S].
2. Select the directory where you want to save the file.
3. Enter the filename in the Name box.
4. From the File Type box, select the format in which you want to save the file.

In all modules you can save your documents as native LibreOffice files, LibreOffice templates, Microsoft files and StarOffice files.

## 4.3.2 Navigator and Navigation Shortcuts

The Navigator tool displays information about the contents of a document. It also lets you quickly jump to different elements. For example, you can use the Navigator to get

a quick overview of all images included in the document. To open the Navigator, click View > Navigator or with [F5]. The elements listed in the Navigator vary according to the LibreOffice module being used.

You can also navigate your files with the following shortcuts:

Table 4.2 Navigation Shortcuts

| Action | Shortcut |
|---|---|
| Delete to the End of the Word | [] + [] |
| Delete to the Beginning of the Word | [] + [] |
| End of Document | [] + [] |
| End of Line | [] |
| Select All | [] + [A] |
| Start of Line | [] |
| Top of Document | [] + [] |
| Word Left | [] + [] |
| Word Right | [] + [] |

### 4.3.3 Using and Creating Templates

LibreOffice comes with a set of predefined templates, and you can find additional templates on the Internet. If you want to create your own templates, you need to determine what you want the document to look like and create the styles you need in that template.

For text document, spreadsheets, presentations, and drawings, you can easily create a template from an existing document as follows:

1 Start LibreOffice and open or create a document that contains the styles that you want to use as a template.

2 Click File > Templates > Save.

3 Specify a name for the template.

4 In the Categories box, click the category where you want to place the template.

5 Click OK.

## 4.4 LibreOffice Writer

LibreOffice Writer is a full-featured word processor with page and text-formatting capabilities. It also includes some features that are usually found only in expensive desktop publishing applications. Many of the LibreOffice Writer features apply also to other LibreOffice modules.

### 4.4.1 Creating a New Document

LibreOffice Writer allows you to create a new document in one of the following ways:

- From Scratch To create a document from scratch, click File > New > Text Document and a new empty Writer document is created.
- Wizard To use a standard format and predefined elements for your own documents use a wizard. Click File > Wizards > Letter... and follow the steps.
- Templates To use a template, click File > New > Templates and Documents and choose one of the many folders (for example, `Business Correspondence`) and a new document based on the style of your selected template is created.

To use a standard format and predefined elements for your own documents, try a wizard (a small utility that lets you make some basic initial decisions and then produces a ready-made document from a template). For example, to create a business letter, click File > Wizards > Letter. With the wizard's dialogs, you can create a basic document that uses a standard format.

Enter text in the document window as desired. Use the Formatting toolbar or the Format menu to adjust the appearance of the document. Use the File menu or the relevant buttons in the toolbar to print and save your document. With the options under Insert, add extra items to your document, such as a table, picture, or chart.

### 4.4.2 Customizing LibreOffice

With LibreOffice you can customize the default settings with the Format menu.

Below is a list of the most popular options available in most modules:

Customizing Character Settings
: To set all character elements at the same time, select Format > Character.

Customizing Paragraph Settings
: To format paragraphs, select Format > Paragraph and select the appropriate tabs for making changes.

Customizing Alignment
: To modify the alignment, select Format > Paragraph and choose the alignment you want.

Inserting Headers and Footers

If you want to add headers and footers, click Insert and select either Header or Footer.

Inserting Special Characters

If you want to insert special characters, select Insert > Special Character.

Inserting Footnotes

If you want to insert footnotes, click Insert and select Footnote.

Inserting Columns

If you want to insert columns, click Insert and select Section. Then in the Columns tab, specify the number of columns to use for the current section.

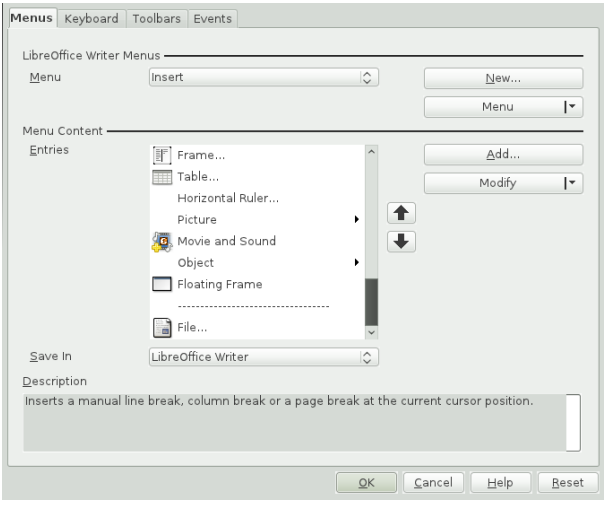
Inserting Indexes and Table of Contents

If you want to insert indexes or a table of contents, click Insert and select Indexes and Tables.

The Object Toolbar offers additional text formatting options.

Inserting Comments

If you want to comment your text for others or for yourself, click Insert > Comment.




## 4.4.3 Using Master Documents

If you are working with a very large document (such as a book), you might find it easier to manage the book with a master document, rather than keeping the book in a single

file. A master document enables you to quickly apply formatting changes to a large document or to jump to each subdocument for editing.

A master document is a Writer document that serves as a container for multiple Writer files. You can maintain chapters or other subdocuments as individual files collected in the master document. Master documents are also useful if multiple people are working on a document. You can separate each person's relevant part of the document into subdocuments, allowing multiple writers to work on their subdocuments at the same time without the danger of overwriting each other's work.

1 Click New > Master Document.

   or

   Open an existing document and click File > Send > Create Master Document.

2 Insert subdocuments.

3 Click File Save.

The LibreOffice help files contain more complete information about working with master documents. Look for the topic entitled Working with Master Documents and Subdocuments.

### 4.4.4 Using Writer as HTML Editor

In addition to being a full-featured word processor, Writer also functions as an HTML editor. Writer includes HTML tags that can be applied the same way as any other style in a Writer document. You can view the document as it will appear online, or you can directly edit the HTML code.

1 Click File > New > HTML Document.

2 Press F11 to open the Styles and Formatting window.

3 Click the arrow at the bottom of the Styles and Formatting window.

4 Select HTML Styles.

5 Create your HTML document, using the styles to tag your text.

6 Click File > Save As.

7 Select the location where you want to save your file, name the file, and select HTML Document (.html) from the File Type list.

8 Click OK.

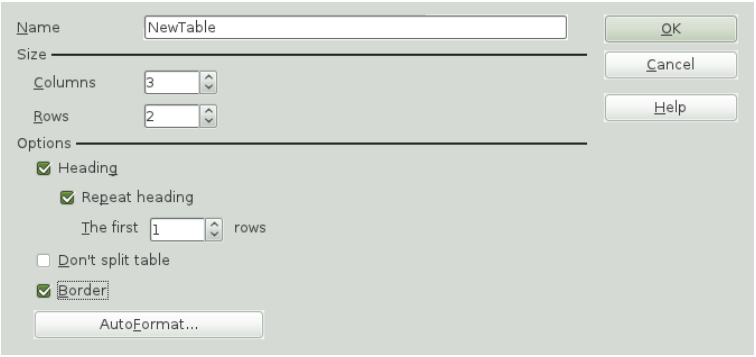
If you prefer to edit HTML code directly, or if you want to see the HTML code created when you edited the HTML file as a Writer document, click View > HTML Source. In

HTML Source mode, the Formatting and Styles list is not available. The first time you switch to HTML Source mode, you are prompted to save the file as HTML, if you have not already done so.

### 4.4.5 Tables

You can include tabular data in the Writer, Impress and Draw module. Insert basic tables directly into your document either with the following menus items or by using the toolbar:

- Insert > Table,
- Table > Insert > Table,
- or use the Table icon.



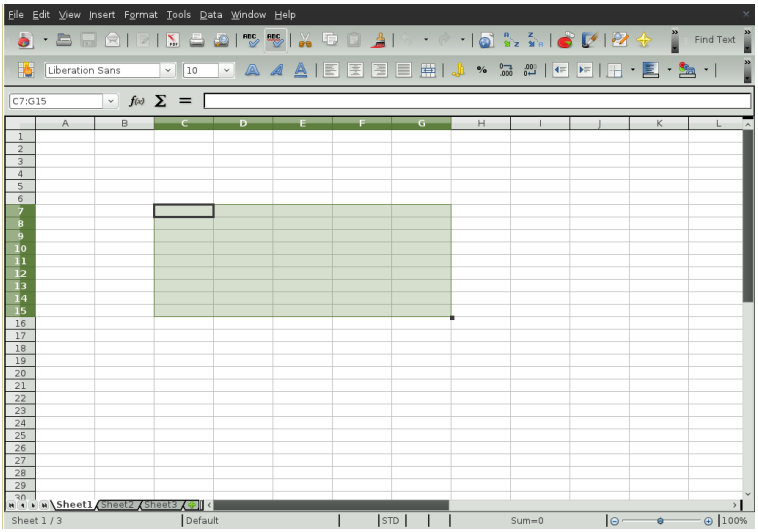

To enter data, select a cell with your mouse pointer and begin typing. To move around cells, use the arrow keys. [] moves to the next cell, skipping over the contents of the cell and [] + [] moves backwards.

You can use Table menu or the Table toolbar to modify the table.

## 4.5 LibreOffice Calc

Calc is the LibreOffice spreadsheet and data-plotting module. Spreadsheets consist of a number of sheets, containing cells which can be filled with elements like text, numbers, or formulas. A formula can manipulate data from other cells to generate a value for the cell into which it is inserted. Calc also allows you to define ranges, filter and sort the data, or create charts from the data to present it graphically. By using DataPilots, you can combine, analyze or compare larger amounts of data.

As with the entire LibreOffice suite, Calc can be used across a variety of platforms. It provides a number of exchange formats (including export to PDF documents), and can also read and save files in Microsoft Excel's format.




## 4.5.1 Creating a New Document

Start LibreOffice and select File > New > Spreadsheet to create a new spreadsheet. Access the individual sheets by clicking the respective tabs at the bottom of the window.

Enter data in the cells as desired. To adjust appearance, either use the Formatting toolbar or the Format menu—or define styles. Use the File menu or the relevant buttons in the toolbar to print and save your document.

## 4.5.2 Customizing LibreOffice Calc

Calc comes with a few built-in cell and page styles to improve the appearance of your spreadsheets and reports. Although these built-in styles are adequate for many uses, you will probably find it useful to create styles for your own frequently-used formatting preferences.

Procedure 4.1 Creating a Style

1. Click Format > Styles and Formatting.
2. In the Styles and Formatting window, click one of the icons representing either Paragraph Styles, Character Styles, or Page Styles icon.
3. Right-click in the Styles and Formatting window, then click New.
4. Specify a name for your style and use the various tabs to set the desired formatting options.

5 Click OK.

Procedure 4.2 Modifying a Style

1 Click Format > Styles and Formatting.

2 In the Styles and Formatting window, click either the Cell Styles or the Page Styles icon.

3 Right-click the name of the style you want to change, then click Modify.

4 Change the desired formatting options.

5 Click OK.

## 4.6 Other LibreOffice Modules

Besides LibreOffice Writer and LibreOffice Calc, LibreOffice also includes the modules Impress, Base, Draw, and Math. With these you can create presentations, design databases, draw up graphics and diagrams and create mathematical formulas.

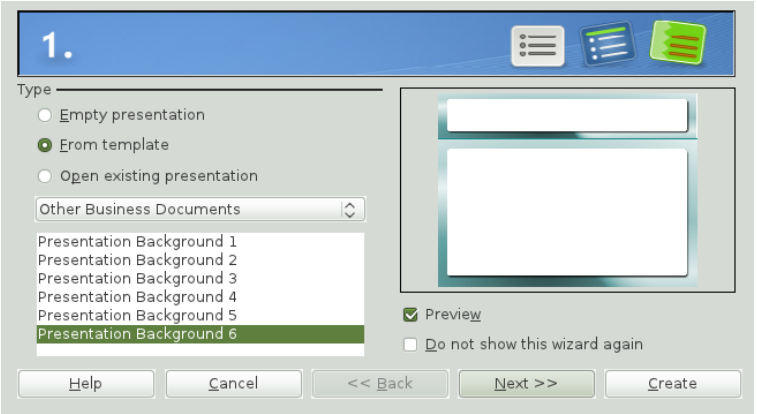
### 4.6.1 Using Presentations with Impress

Use LibreOffice Impress to create presentations for screen display or printing, such as slide shows or transparencies. If you have used other presentation software, you can move comfortably to Impress, which works very similarly to other presentation software.

Impress can open and save Microsoft Powerpoint presentations, which means you can exchange presentations with Powerpoint users, as long as you save your presentations in Powerpoint format.

LibreOffice allows you to use 3D transitions for your slides. Access the available transitions by selecting Slide Show > Slide Transitions.

You can either create a brand new presentation (without any preformatted slides) or use an existing template or presentation for your new document. Impress uses styles and templates the same way other LibreOffice modules do. A wizard leads you through the options available for creating new presentations.




## 4.6.2 Using Databases with Base

LibreOffice includes a database module (Base). Use Base to design a database to store many different kinds of information, from a simple address book or recipe file to a sophisticated document management system.

Tables, forms, queries, and reports can be created manually or using convenient wizards. For example, the Table Wizard contains a number of common fields for business and personal use. A wizard guides you through the steps to create a new database.

LibreOffice Base comes with several predefined database fields to help you create a database. A wizard guides you through the steps to create a new database. The following steps are specific to creating an address book using predefined fields, but it should be easy to use the predefined fields for any of the built-in database options. The process for creating a database can be broken into several subprocesses:

Procedure 4.3 Creating a Database

1 Click File > New > Database.

2 Select Create a new database Next.

3 Click Yes, register the database for me to make your database information available to other LibreOffice modules, select both check boxes in the bottom half of the dialog, then click Finish.

4 Browse to the directory where you want to save the database, specify a name for the database, then click OK.

Procedure 4.4 Setting Up the Database Table

1 In the Table Wizard, click Personal.

2 The Sample tables list changes to show the predefined tables for personal use. If you had clicked Business, the list would contain predefined business tables.

In the Sample tables list, click Addresses. The available fields for the predefined address book appear in the Available fields menu.

3 In the Available fields menu, click the fields you want to use in your address book.

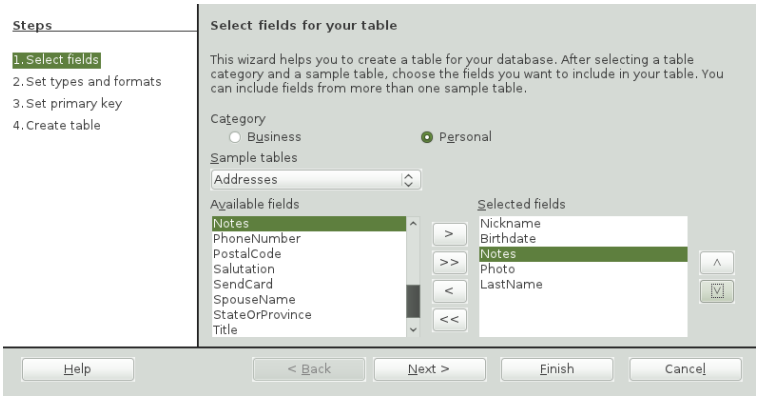
You can select one item at a time, or you can shift-click multiple items to select them.

4 Click the single arrow icon to move the selected items to the Selected fields menu.

To move all available fields to the Selected fields menu, click the double right-arrow.

5 Use the arrow keys to adjust the order of the selected fields.

The fields appear in the table and forms in the order in which they are listed.




## 4.6.3 Creating Graphics with Draw

Use LibreOffice Draw to create graphics and diagrams. You can save your drawings in today's most common formats and import them into any application that lets you import graphics, including the other LibreOffice modules. You can also create Flash versions of your drawings.

LibreOffice allows you to import Scalable Vector Graphics (*.svg) and WordPerfect Graphics (*.wpg). Select File > Open and use the Filter drop-down list to view which file formats can be opened.

Procedure 4.5 Creating a Graphic

1. Click [] + [F2] and enter `oodraw` to open LibreOffice Draw.
2. Use the toolbar at the bottom of the window to create a graphic.
3. Save the graphic.

To embed an existing Draw graphic into an LibreOffice document, select Insert > Object > OLE Object. Select Create from file and click Search to navigate to the Draw file to insert. If you insert a file as OLE object, you can easily edit the object later by double-clicking it.

Procedure 4.6 Opening Draw From Other LibreOffice Modules

One particularly useful feature of Draw is the ability to open it from other LibreOffice modules so you can create a drawing that is automatically imported into your document.

1. From an LibreOffice module (for example, from Writer), click Insert > Object > OLE Object > LibreOffice 3.x Drawing > OK. This opens Draw.
2. Create your drawing.
3. Click in your document, outside the Draw frame.

   The drawing is automatically inserted into your document.

### 4.6.4 Creating Mathematical Formulas with Math

It is usually difficult to include complex mathematical formulas in your documents. The LibreOffice Math equation editor lets you create formulas using operators, functions, and formatting assistants. You can then save those formulas as objects that can be imported into other documents. Math functions can be inserted into other LibreOffice documents like any other graphic object.

Using Math, you can enter your equation in three different ways:

- Type markup in the equation editor,
- Right-click on the equation editor and select the symbol from the context menu,

  or

- Select a symbol from the Selection toolbox.

---

NOTE

Math is not a calculator. The functions it creates are graphical objects. Even if they are imported into Calc, these functions cannot be evaluated.

---

## 4.7 For More Information

LibreOffice also contains extensive online help. In addition, a large community of users and developers support it. For more details, see the following links:

http://www.libreoffice.org/download/3-4-new-features-and-fixes/
A list of all new features available with LibreOffice.

http://www.libreoffice.org/get-help/documentation/
Official LibreOffice support page, provides manuals and other documentation

http://www.taming-openoffice-org.com/
Taming LibreOffice: Books, news, tips and tricks.

# Part II. Managing and Updating Software

Use YaST's software management tool to search for software components you want to add or remove. YaST resolves all dependencies for you. To install packages not shipped with the installation media, add additional software repositories to your setup and let YaST manage them. Keep your system up-to-date by managing software updates with the update applet.

Change the software collection of your system with YaST Software Manager. This YaST module is available in three toolkit flavors: Qt (for KDE desktops), GTK+ (for GNOME desktops), and ncurses (providing a pseudo-graphical user interface in text mode). This chapter describes Qt and GTK+ flavors—for details on the ncurses YaST, see Chapter 12, YaST in Text Mode (page 147).

---

TIP: Changing the Toolkit Flavor

By default, YaST is started with the toolkit matching your desktop (GTK+ under GNOME, Qt under KDE). To alter this default setting system-wide, change the variable `WANTED_GUI` in `/etc/sysconfig/yast2` to either `qt` or `gtk`.

If you do not want to change the system-wide settings, you can nevertheless start YaST in the desired flavor from command line by using the `--gtk` or `--qt`. For example: `yast2 --gtk`.

---

NOTE: Confirmation and Review of Changes

When installing, updating or removing packages, any changes in the Software Manager are not applied immediately but only after confirming them with Accept or Apply respectively. YaST maintains a list with all actions, allowing you to review and modify your changes before applying them to the system.

---

## 5.1 Definition of Terms

Repository
: A local or remote directory containing packages, plus additional information about these packages (package meta-data).

(Repository) Alias
: A short name for a repository used by various zypper commands. The alias can be chosen by the user when adding a repository and must be unique.

Product
: Represents a whole product, for example openSUSE®.

Pattern
: A pattern is an installable group of packages dedicated to a certain purpose. For example, the `Laptop` pattern contains all packages that are needed in a mobile computing environment. Patterns define package dependencies (such as required or recommended packages) and come with a preselection of packages marked for installation. This ensures that the most important packages needed for a certain purpose are available on your system after installation of the pattern. However, not necessarily all packages in a pattern are preselected for installation and you can manually select or deselect packages within a pattern according to your needs and wishes.

Package
: A package is a compressed file in `rpm` format that contains the files for a particular program.

Patch
: A patch consists of one or more packages and may be applied by means of deltarpms. It may also introduce dependencies to packages that are not installed yet.

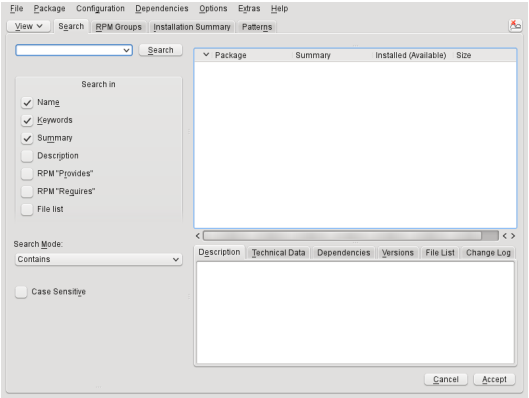
Resolvable
: An generic term for product, pattern, package or patch. The most commonly used type of resolvable is a package or a patch.

deltarpm
: A deltarpm consists only of the binary diff between two defined versions of a package, and therefore has the smallest download size. Before being installed, the full RPM package is rebuilt on the local machine.

Package Dependencies
: Certain packages are dependent on other packages, such as shared libraries. In other terms, a package may `require` other packages—if the required packages are not available, the package cannot be installed. In addition to dependencies (package requirements) that must be fulfilled, some packages `recommend` other packages. These recommended packages are only installed if they are actually available, otherwise they are just ignored and the package recommending them is installed nevertheless.

## 5.2 Using the KDE Interface (Qt)

The YaST Qt interface is started by default when using the desktops KDE, icewm, and others. It is also used when invoking YaST from a remote terminal. Start the software manager from the YaST Control Center by choosing Software > Software Management.




## 5.2.1 Views for Searching Packages or Patterns

The YaST software manager can install packages or patterns from all currently enabled repositories. It offers different views and filters to make it easier to find the software you are searching for. The Search  view is the default view of the window. To change view, click View and select one of the following entries from the drop-down list. The selected view opens in a new tab.

Patterns
: Lists all patterns available for installation on your system.

Package Groups
: Lists all packages sorted by groups such as Graphics, Programming, or Security.

RPM Groups
: Lists all packages sorted by functionality with groups and subgroups. For example Networking > Email > Clients.

Languages
: Filter to list all packages needed to add a new system language.

Repositories
: Filter to list packages by repository. In order to select more than one repository, hold the [] key while clicking on repository names. The “pseudo repository” @System lists all packages currently installed.

Search
: Lets you search for a package according to certain criteria. Enter a search term and press []. Refine your search by specifying where to Search In and by changing the Search Mode. For example, if you do not know the package name but only the name of the application that you are searching for, try including the package Description in the search process.

Installation Summary
: If you have already selected packages for installation, update or removal, this view shows the changes that will be applied to your system as soon as you click Accept. To filter for packages with a certain status in this view, activate or deactivate the respective check boxes. Hit [] + [F1] for details on the status flags.

---

TIP: Finding Packages not Belonging to an Active Repository

To list all packages that do not belong to an active repository, choose View > Repositories > @System and then choose Secondary Filter > Unmaintained Packages. This is useful, for example, if you have deleted a repository and would like to make sure no packages from that repository remain installed.

---

## 5.2.2 Installing and Removing Packages or Patterns

Certain packages are dependent on other packages, such as shared libraries. On the other hand, some packages cannot coexist with others on the system. If possible, YaST automatically resolves these dependencies or conflicts. If your choice results in a dependency conflict that cannot be automatically solved, you need to solve it manually as described in Section 5.2.4, “Checking Software Dependencies” (page 83).

---

NOTE: Removal of Packages

When removing any packages, by default YaST only removes the selected packages. If you want YaST to also remove any other packages that become unneeded after removal of the specified package, select Options > Cleanup when deleting packages.

---

1. Search for packages as described in Section 5.2.1, “Views for Searching Packages or Patterns” (page 79).

2. The packages found are listed in the right pane. To install a package or remove it, right-click it and choose Install or Delete. If the relevant option is not available, check the package status indicated by the symbol in front of the package name —hit [] + [F1] for help.

   ---

   TIP: Applying an Action to All Packages Listed

   To apply an action to all packages listed in the right pane, choose an action from Package > All in This List.

   ---

3. To install a pattern, right-click the pattern name and choose Install.

4. It is not possible to remove a pattern per se. Instead, select the packages of a pattern you want to remove and mark them for removal.

5. In order to select more packages, repeat the steps mentioned above.

6 Before applying your changes, you can review or modify them by clicking View > Installation Summary. By default, all packages that will change status, are listed.

7 In order to revert the status for a package, right-click the package and select one of the following entries: Keep if the package was scheduled to be deleted or updated, or Do Not Install if it was scheduled for installation. To abandon all changes and close the Software Manager, click Cancel and Abandon

8 When you are finished, click Accept to apply your changes.

9 In case YaST found dependencies on other packages, a list of packages that have additionally been chosen for installation, update or removal is presented. Click Continue to accept them.

After all selected packages are installed, updated or removed, the YaST Software Manager automatically terminates.

---

NOTE: Installing Source Packages

Installing source packages with YaST Software Manager is not possible at the moment. Use the command line tool `zypper` for this purpose. For more information, see Section 9.1.2.1, “Installing Source Packages” (page 111).

---

## 5.2.3 Updating Packages

Instead of updating individual packages, you can also update all installed packages or all packages from a certain repository. When mass updating packages, the following aspects are generally considered:

- priorities of the repositories that provide the package,
- architecture of the package (for example, x86_64, i686, i586),
- version number of the package,
- package vendor.

Which of the aspects has the highest importance for choosing the update candidates depends on the respective update option you choose.

1 To update all installed packages to the latest version, choose Package > All Packages > Update if Newer Version Available from the main menu.

All repositories are checked for possible update candidates, using the following policy: YaST first tries to restrict the search to packages with the same architecture and vendor like the installed one. If the search is positive, the “best” update candidate from those is selected according to the process below. However, if no comparable package of the same vendor can be found, the search is expand-

ed to all packages with the same architecture. If still no comparable package can be found, all packages are considered and the “best” update candidate is selected according to the following criteria:

1. Repository priority: Prefer the package from the repository with the highest priority.

2. If more than one package results from this selection, choose the one with the “best” architecture (best choice: matching the architecture of the installed one; otherwise: x86_64 > i686 > i586).

If the resulting package has a higher version number than the installed one, the installed package will be updated and replaced with the selected update candidate.

This option tries to avoid changes in architecture and vendor for the installed packages, but under certain circumstances, they are tolerated.

---

NOTE: Update Unconditionally

If you choose Package > All Packages > Update Unconditionally instead, basically the same criteria apply but candidate package found is installed unconditionally. Thus, choosing this option might actually lead to downgrading some packages.

---

2 To make sure that the packages for a mass update derive from a certain repository:

2a Choose the repository from which to update as described in Section 5.2.1, “Views for Searching Packages or Patterns” (page 79) .

2b On the right hand side of the window, click Switch system packages to the versions in this repository. This explicitly allows YaST to change the package vendor when replacing the packages.

As soon as you proceed with Accept, all installed packages will be replaced by packages deriving from this repository, if available. This may lead to changes in vendor and architecture and even to downgrading some packages.

2c To refrain from this, click Cancel switching system packages to the versions in this repository. Note that you can only cancel this until you press the Accept button.

3 Before applying your changes, you can review or modify them by clicking View > Installation Summary. By default, all packages that will change status, are listed.

4 If all options are set according to your wishes, confirm your changes with Accept to start the mass update.

## 5.2.4 Checking Software Dependencies

Most packages are dependent on other packages. If a package, for example, uses a shared library, it is dependent on the package providing this library. On the other hand some packages cannot coexist with each other, causing a conflict (for example, you can only install one mail transfer agent: sendmail or postfix). When installing or removing software, the Software Manager makes sure no dependencies or conflicts remain unsolved to ensure system integrity.

In case there exists only one solution to resolve a dependency or a conflict, it is resolved automatically. Multiple solutions always cause a conflict which needs to be resolved manually. If solving a conflict involves a vendor or architecture change, it also needs to be solved manually. When clicking Accept to apply any changes in the Software Manager, you get an overview of all actions triggered by the automatic resolver which you need to confirm.

By default, dependencies are automatically checked. A check is performed every time you change a package status (for example, by marking a package for installation or removal). This is generally useful, but can become exhausting when manually resolving a dependency conflict. To disable this function, uncheck Dependencies > Autocheck. Manually perform a dependency check with Dependencies > Check Now. A consistency check is always performed when you confirm your selection with Accept.

To review a package's dependencies, right-click it and choose Show Solver Information. A map showing the dependencies opens. Packages that are already installed are displayed in a green frame.

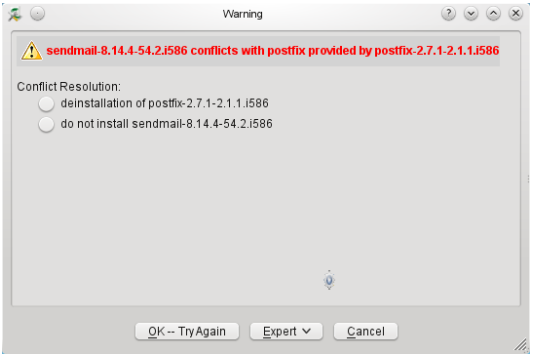
---

NOTE: Manually Solving Package Conflicts

Unless you are very experienced, follow the suggestions YaST makes when handling package conflicts, otherwise you may not be able to resolve them. Keep in mind that every change you make, potentially triggers other conflicts, so you can easily end up with a steadily increasing number of conflicts. In case this happens, Cancel the Software Manager, Abandon all your changes and start again.

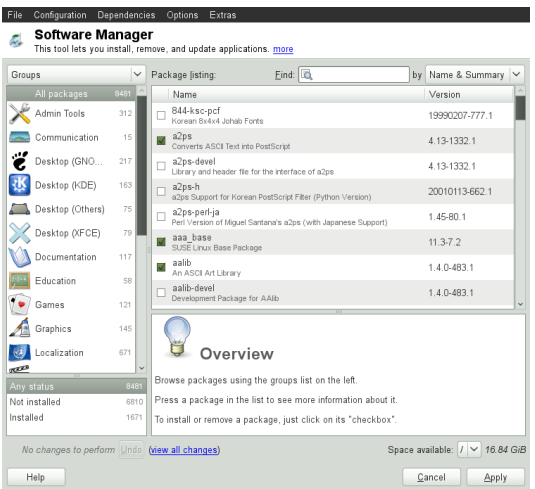
---

Figure 5.1 Conflict Management of the Software Manager




# 5.3 Using the GNOME Interface (GTK+)

The YaST GTK+ interface is started by default when using the desktops GNOME and XFCE. Start the software manager from the YaST Control Center by clicking Software > Software Management.




## 5.3.1 Views for Searching Packages or Patterns

The easiest way to find a package is to use the search field in the upper right corner of the software manager. Enter a search term and press []. By default it will search

package names and summaries. Press the search item to change this filter and search the file lists, for example.

The software manager also offers different views and filters for displaying package lists. These are available from the pull-down menu in the upper left corner:

Groups
: The default view lists all packages sorted by groups such as Admin Tools, Graphics, Programming, or Security.

RPM Groups
: Lists all packages sorted by functionality with groups and subgroups. For example Networking > Email > Clients.

Repositories
: Filter to list packages by repository. In order to select more than one repository, hold the [] key while clicking on repository names. The “pseudo repository” @System lists all packages currently installed.

  To add, edit, or remove available repositories, click Edit Repositories.

Patterns
: Lists all patterns available for installation on your system.

Languages
: Filter to list all packages needed to add a new system language.

The box in the lower right corner of the dialog also allows to filter for packages that are Installed, Not Installed or Upgradable. If you select the No Status entry, all available packages from the configured repositories are displayed, independent of their status.

## 5.3.2 Installing and Removing Packages or Patterns

Certain packages are dependent on other packages, such as shared libraries. On the other hand, some packages cannot coexist with others on the system. If possible, YaST automatically resolves these dependencies or conflicts. If your choice results in a dependency conflict that cannot be automatically solved, you need to solve it manually as described in Section 5.2.4, “Checking Software Dependencies” (page 83).

1. Search for packages as described in Section 5.3.1, “Views for Searching Packages or Patterns” (page 84).

2. The packages found are listed in the right pane. To further filter the search results according to package status (Any Status, Not Installed, Installed, Upgradable), select one of the entries in the box at the lower left corner of the dialog. For details about a package, click the package in the list. Information like available versions, authors and changelog of the package are displayed in the lower right corner of the window.

   To mark a package for installation, re-installation, removal, or upgrade, right-click the package and choose the appropriate action from the menu.

TIP: Applying an Action to All Packages Listed

To apply an action to all packages listed in the right pane, right-click a package, choose Select All, right-click again and choose an action.

3 To install a pattern, right-click the pattern name and choose Install.

4 It is not possible to remove a pattern per se. Instead, select the packages of a pattern you want to remove and mark them for removal.

5 In order to select more packages, repeat the steps mentioned above.

6 Before applying your changes, you can review or modify them by clicking View All Changes at the bottom of the dialog. By default, all packages that will change status are listed.

7 To revert changes for a package, click the Undo icon with the yellow arrow. To finish the review, click Close.

8 When you are finished with the selection of packages to install or remove, Apply your changes.

9 In case YaST found dependencies on other packages, a list of packages that have additionally been chosen for installation, update or removal is presented. Click Apply to accept them.

After all selected packages are installed, updated or removed, the YaST Software Manager automatically terminates.

NOTE: Installing Source Packages

Installing source packages with YaST Software Manager is not possible at the moment. Use the command line tool `zypper` for this purpose. For more information, see Section 9.1.2.1, “Installing Source Packages” (page 111).

## 5.3.3 Updating Packages

Instead of updating individual packages, you can also update all installed packages or all packages from a certain repository. When mass updating packages, the following aspects are generally considered:

- priorities of the repositories that provide the package,
- architecture of the package (for example, x86_64, i686, i586),
- version number of the package,
- package vendor.

Which of the aspects has the highest importance for choosing the update candidates depends on the respective update option you choose.

1 To view the list of packages that can be updated (packages with higher versions are available), select Upgradable in the bottom left box.

2 To update all packages listed there, click Upgrade All.

   To install only upgradable packages for which an official patch has been issued, click Upgrade Patches. Those packages are marked by a patch tag next to their version number. Choosing this option is equivalent to doing an online update with YaST as described in Chapter 6, YaST Online Update (page 97).

   If no patches have been issued since last applying patches, the button is disabled.

3 To make sure that the packages for a mass update derive from a certain repository:

   3a Choose the repository from which to update as described in Section 5.3.1, “Views for Searching Packages or Patterns” (page 84) .

   3b On the right hand side of the window, click Switch system packages to the versions in this repository. This explicitly allows YaST to change the package vendor when replacing the packages.

      All installed packages will be replaced by packages deriving from this repository, if available. This may lead to changes in vendor and architecture and even to downgrading some packages.

4 Before applying the changes, you can review or modify them by clicking View All Changes at the bottom of the dialog. By default, all packages that will change status are listed.

5 To refrain from switching the system packages to the versions in this repository, click the Undo button next to the respective option.

6 If all options are set according to your wishes, confirm your changes with Apply to start the mass update.

### 5.3.4 Checking Software Dependencies

Most packages are dependent on other packages. If a package, for example, uses a shared library, it will be dependent on the package providing this library. On the other hand, some packages cannot coexist with each other, causing a conflict (for example, you can only install one mail transfer agent: sendmail or postfix). When installing or removing software, the Software Manager makes sure no dependencies or conflicts remain unresolved to ensure system integrity.

In case there exists only one solution to resolve a dependency or a conflict, it is resolved automatically. Multiple solutions always cause a conflict which needs to be re-

solved manually. If solving a conflict involves a vendor or architecture change, it also needs to be solved manually. When clicking Apply to apply any changes in the Software Manager, you get an overview of all actions triggered by the automatic resolver which you need to confirm.

By default, dependencies are automatically checked. A check is performed every time you change a package status (for example, by marking a package for installation or removal). This is generally useful, but can become exhausting when manually resolving a dependency conflict. To disable this function, uncheck Dependencies > Autocheck. Manually perform a dependency check with Dependencies > Check Now. A consistency check is always performed when you confirm your selection with Apply.

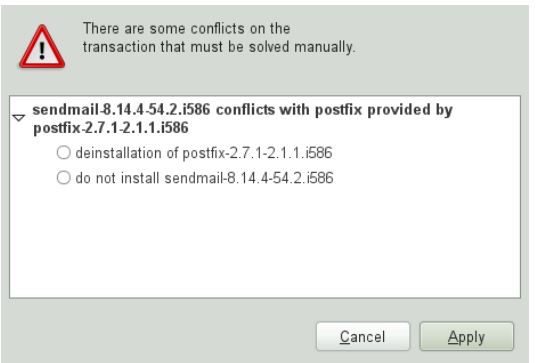
---

NOTE: Manually Solving Package Conflicts

Unless you are very experienced, follow the suggestions YaST makes when handling package conflicts, otherwise you may not be able to resolve them. Keep in mind that every change you make, potentially triggers other conflicts, so you can easily end up with a steadily increasing number of conflicts. In case this happens, click Cancel and Quit the software manger. Relaunch it to start again.

---

Figure 5.2 Conflict Management of the Software Manager




## 5.4 Managing Software Repositories and Services

If you want to install third-party software, add additional software repositories to your system. By default, the product repositories such as openSUSE-DVD 12.2 and a matching update repository are automatically configured during the installation. Depending on the initially selected product, a separate language add-on repository with translations, dictionaries, etc. might also be configured.

To manage repositories, start YaST and select Software > Software Repositories. The Configured Software Repositories dialog opens. Here, you can also manage subscriptions to so-called Services by changing the View at the right corner of the dialog to All

Services. A Service in this context is a Repository Index Service (RIS) that can offer one or more software repositories. Such a Service can be changed dynamically by its administrator or vendor.

Each repository provides files describing content of the repository (package names, versions, etc.). These repository description files are downloaded to a local cache that is used by YaST. To ensure their integrity, software repositories can be signed with the GPG Key of the repository maintainer. Whenever you add a new repository, YaST offers the ability to import its key.

---

WARNING: Trusting External Software Sources

Before adding external software repositories to your list of repositories, make sure this repository can be trusted. openSUSE is not responsible for any potential problems arising from software installed from third-party software repositories.

---

### 5.4.1 Adding Software Repositories

You can either add repositories from a local harddisk, from a removable medium (like a CD, DVD or a USB mass storage) or from a network.

To add repositories from the Configured Software Repositories dialog in YaST proceed as follows:

1. Click Add.

2. From the list of available Media Types specify the type matching your repository:

   For network sources, it is usually sufficient to use the default option, Specify URL.

   To add a repository from a removable medium or a local harddisk, choose the relevant option and insert the medium or connect the USB device to the machine, respectively.

3. You can choose to Download Repository Description Files now. If the option is unchecked, YaST will automatically download the files later, if needed. Click Next to proceed.

4. When adding a repository from the network, enter the data you are prompted for. Continue with Next.

5. Depending on the repository you have added, you might be asked if you want to import the GPG key with which it is signed or asked to agree to a license.

   After confirming these messages, YaST will download and parse the metadata and add the repository to the list of Configured Repositories..

6. If needed, adjust the repository Properties as described in Section 5.4.2, “Managing Repository Properties” (page 90) or confirm your changes with OK to close the configuration dialog.

Now you can install software from this repository as described in Section 5.2, “Using the KDE Interface (Qt)” (page 78) or in Section 5.3, “Using the GNOME Interface (GTK+)” (page 84).

YaST also offers a list of predefined popular repositories available under Community Repositories. Among others, it includes driver repositories for nVidia and ATI graphics cards, and popular projects from the openSUSE® Build Service, such as the Mozilla repository (containing packages with the most recent versions of Firefox and Thunderbird).

Procedure 5.1 Configuring Community Repositories

1. Start YaST and select Software > Software Repositories.
2. Click Add.
3. Choose Community Repositories and proceed with Next.
4. From the preconfigured list of repositories choose the ones you want to add by ticking the respective check boxes. The Mozilla repository, for example, is listed as openSUSE BuildService - Mozilla.

   Confirm with OK.
5. Accept to Import the GPG key. You need to import a key for each repository you have chosen.
6. The new software repositories are now listed in the Configured Software Repositories overview. Click OK to leave the software repositories configuration.

## 5.4.2 Managing Repository Properties

The Configured Software Repositories overview of the Software Repositories lets you change the following repository properties:

Status
: The repository status can either be Enabled or Disabled. You can only install packages from repositories that are enabled. To turn a repository off temporarily click Disable. You can also double-click on a repository name to toggle its status. If you want to remove a repository completely, click Delete.

Refresh
: When refreshing a repository, its content description (package names, versions, etc.) is downloaded to a local cache that is used by YaST. It is sufficient to do this once for static repositories such as CDs or DVDs, whereas repositories whose content changes often should be refreshed frequently. The easiest way to keep a repository's cache up-to-date is to choose Automatically Refresh. To do a manual refresh click Refresh and select one of the options.

Keep Downloaded Packages
: Packages from remote repositories are downloaded before being installed. By default, they are deleted upon a successful installation. Activating Keep Downloaded

Packages prevents the deletion of downloaded packages. The download location is configured in `/etc/zypp/zypp.conf`, by default it is `/var/cache/zypp/packages`.

Priority
: The Priority of a repository is a value between `1` and `200`, with `1` being the highest priority and `200` the lowest priority. Any new repositories that are added with YaST get a priority of `99` by default. If you do not care about a priority value for a certain repository, you can also set the value to `0` to apply the default priority to that repository (`99`). If a package is available in more than one repository, then the repository with the highest priority takes precedence. This is useful if you want to avoid downloading packages unnecessarily from the Internet by giving a local repository (for example, a DVD) a higher priority.

---

IMPORTANT: Priority vs. Version

The repository with the highest priority takes precedence in any case. Therefore, make sure that the update repository always has the highest priority (20 by default), otherwise you might install an outdated version that will not be updated until the next online update.

If you add repositories providing new versions for programs shipped with openSUSE (for example a repository with the latest KDE or GNOME version), make sure they have a higher priority than the standard repositories, otherwise packages from these repositories will not be installed by default.

---

Name and URL
: To change a repository name or its URL, select it from the list with a single-click and then click Edit.

### 5.4.3 Managing Repository Keys

To ensure their integrity, software repositories can be signed with the GPG Key of the repository maintainer. Whenever you add a new repository, YaST offers to import its key. Verify it as you would do with any other GPG key and make sure it does not change. If you detect a key change, something might be wrong with the repository. Disable the repository as an installation source until you know the cause of the key change.

To manage all imported keys, click GPG Keys... in the Configured Software Repositories dialog. Select an entry with the mouse to show the key properties at the bottom of the window. Add, Edit or Delete keys with a click on the respective buttons.

## 5.5 Keeping the System Up-to-date

openSUSE offers a continuous stream of software security patches for your product. The update applet informs you about the availability of patches and lets you easily install them with just a few clicks.

### 5.5.1 Using the KDE Software Updater

The Software Updater icon resides in the system tray of your panel depicting a gear-wheel with a green arrow. To start Software Updater manually, choose System Settings > SoftwareManagement > Software Updates from the main menu. Alternatively, [] + [F2] and enter `kpk_update`.

---

NOTE: Icon Visibility

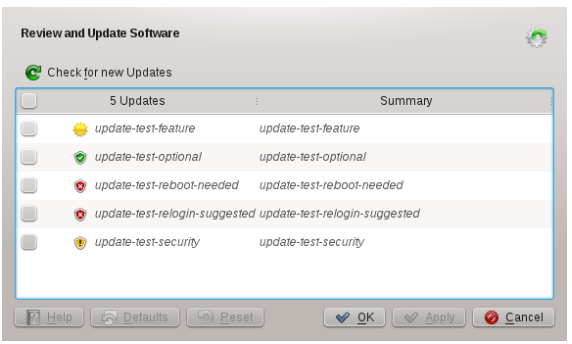
The Software Updater icon is only visible in the system tray, if patches are available. Hover over the icon to see the number of patches available.

---

#### 5.5.1.1 Installing Patches

1. Whenever software updates are available, the applet icon appears in the panel. Left-click the Software Updater icon to launch the Review and Update software window.
2. Select a patch for installation by ticking its checkbox. Get detailed information on a patch by clicking on its title. To select all available patches for installation, tick the checkbox in the table header.
3. Click Apply to start the patch installation.
4. In case you have started the patch installation for the first time, you will be asked to enter the `root` password twice in order to proceed. If you also check Remember authorization you will never be asked again to provide the password.
5. The Additional Changes window showing an installation summary opens. Click Continue to finish the installation.

Figure 5.3 KDE Software Updater




The YaST Online Update offers advanced features to customize the patch installation. Please refer to Chapter 6, YaST Online Update (page 97) for more information.

#### 5.5.1.2 Configuring the KDE Software Updater

By default Software Updater checks for updates every 24 hours, notifies you when patches are available and does not automatically install patches. These settings can be changed with the Software Management settings. To open the Software Management settings choose System Settings > Software Management > Settings from the main menu. Alternatively, press [] + [F2] and enter `kpk_settings`. The settings for Software Updater are available in the Update Settings section.

---

IMPORTANT: Patch Origin

The Software Management settings also allows you to configure the repositories (Origin of Packages) to be used. This setting not only applies to Software Updater but also to the KDE Software Management module (Get and Remove Software).

Make sure the repository Updates for openSUSE 12.2 is always selected—otherwise you will not receive patches.

---

### 5.5.2 Using the GNOME Update Applet

The update applet resides in the notification area of the panel. Its icon changes depending on the availability and relevance of patches and the status of the update. To invoke the applet manually, choose Computer > More Applications > System > Software Update.

---

NOTE: Icon visibility

By default, the update applet icon is only visible in the notification area, if patches are available.

---

Open box with a globe
: The update applet is busy (for example checking for updates or installing software).

Red Star with Exclamation Mark
: Security patches are available.

Orange Star with an Up Arrow
: Important patches are available.

Yellow Star with a Down Arrow
: Trivial patches are available.

Yellow Triangle with Exclamation Mark
: An error has occurred.

#### 5.5.2.1 Installing Patches

Procedure 5.2 Installing Patches

1 Whenever new patches are available, a notification message will appear and the Update Applet icon will be visible in the notification area. Either click Install

updates in the notification message or click the icon to open the Software Update window.

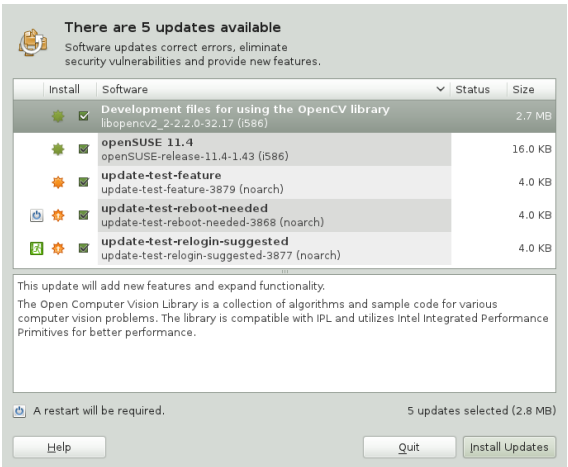
2 All security and important patches are preselected. It is strongly recommended to install these patches. Trivial patches can be manually selected by ticking the respective check boxes. Get detailed information on a patch by clicking on its title.

3 Click Install Updates to start the patch installation.

4 The Additional Confirmation Required window showing an installation summary opens. Click Continue to proceed.

5 Enter the `root` password in the authentication screen and proceed with Authenticate.

Figure 5.4 GNOME Update Applet




The YaST Online Update offers advanced features to customize the patch installation. Please refer to Chapter 6, YaST Online Update (page 97) for more information.

### 5.5.2.2 Configuring the Software Update Applet

To configure the update applet, right-click the update icon in the panel and choose Preferences. The configuration dialog lets you modify the following settings:

Check for Updates
: Choose how often a check for updates is performed: Hourly, Daily, Weekly, or Never.

Automatically Install
: Configure whether patches are installed automatically or not (default). Automatic installation can be chosen for either security patches only or for all patches.

Check for Major Upgrades
: Choose how often a check for major upgrades is performed: Daily, Weekly, or Never.

Check for updates when using mobile broadband
: This configuration option is only available on mobile computers. Turned off by default.

More options are configurable using `gconf-editor`: apps > gnome-packagekit.

openSUSE 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 Section 5.5, “Keeping the System Up-to-date” (page 91) for further information on the update applet. This chapter covers the alternative tool for updating software packages: YaST Online Update.

The current patches for openSUSE® are available from an update software repository, which is automatically configured during the installation. 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 Section 5.4, “Managing Software Repositories and Services” (page 88).

openSUSE provides updates with different relevance levels:

Security Updates
: Fix severe security hazards and should definitely be installed.

Recommended Updates
: Fix issues that could compromise your computer.

Optional Updates
: Fix non-security relevant issues or provide enhancements.

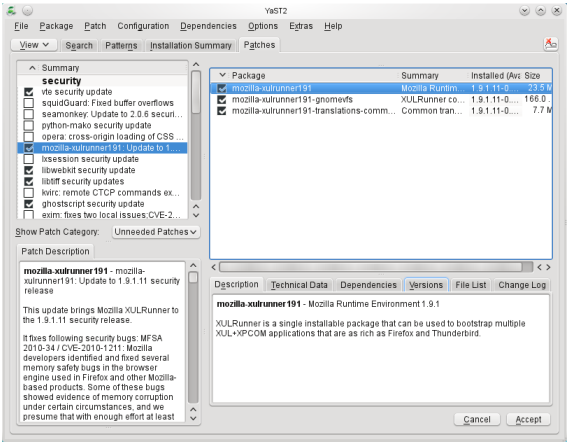
## 6.1 The Online Update Dialog

The YaST Online Update dialog is available in two toolkit flavors: GTK (for GNOME) and Qt (for KDE). Both interfaces differ in look and feel but basically provide the same functions. The following sections provide a brief description of each. To open the dialog, start YaST and select Software > Online Update. Alternatively, start it from the command line with `yast2 online_update`.

### 6.1.1 KDE Interface (Qt)

The Online Update window consists of four sections.

Figure 6.1 YaST Online Update—Qt Interface




The Summary section on the left lists the available patches for openSUSE. 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 have been fulfilled (because the relevant packages have already been updated from another source).

All Patches
: All patches available for openSUSE.

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 [] + [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.

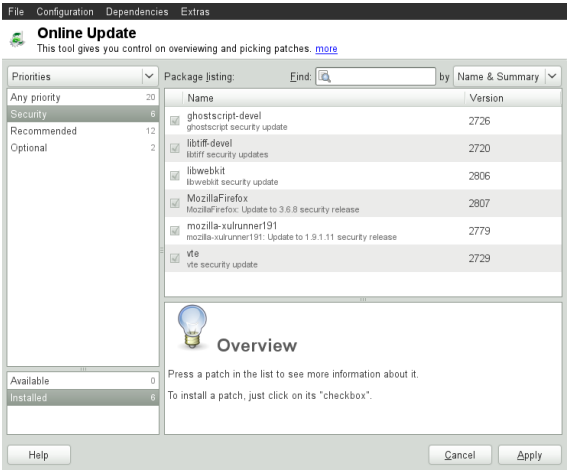
Select an entry in the Summary section to view a short Patch Description at the bottom left corner of the dialog. The upper right section lists the packages included in the

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.

## 6.1.2 GNOME Interface (GTK)

The Online Update window consists of four main sections.

Figure 6.2 YaST Online Update—GTK Interface




The upper right section lists the available (or already installed) patches for openSUSE. To filter patches according to their security relevance, click the corresponding Priority entry in the upper right section of the window: `Security`, `Recommended`, `Optional` or `Any priority`.

If all available patches are already installed, the Package listing in the upper right section will show no entries. The box in the bottom left-hand section shows the number of both available and already installed patches and lets you toggle the view to either Available or Installed patches.

Select an entry in the Package listing section to view a patch description and further details at the bottom right corner of the dialog. As a patch can consist of several packages, click the Applies to entry in the lower right section to see which packages are included in the respective patch.

Click on a patch entry to open a row with detailed information about the patch in the bottom of the window. Here you can see a detailed patch description as well as the versions available. You can also choose to Install optional patches—security and recommended patches are already preselected for installation.

## 6.2 Installing Patches

The YaST Online Update dialog allows you to either install all available patches at one go 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 the `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.

Procedure 6.1 Applying Patches with YaST Online Update

1 Start YaST and select Software > Online Update.

2 To automatically apply all new patches (except the `optional` ones) that are currently available for your system, proceed with Apply or Accept to start the installation of the preselected patches.

3 To first modify the selection of patches that you want to apply:

3a Use the respective filters and views the GTK and Qt interfaces provide. For details, refer to Section 6.1.1, “KDE Interface (Qt)” (page 97) and Section 6.1.2, “GNOME Interface (GTK)” (page 99).

3b Select or deselect patches according to your needs and wishes by activating or deactivating the respective checkbox (GNOME) or by right-clicking the patch and choosing the respective action from the context menu (KDE).

---

IMPORTANT: Always Apply Security Updates

However, do not deselect any `security`-related patches if you do not have a very good reason for doing so. They fix severe security hazards and prevent your system from exploits.

---

3c 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 (KDE).

3d To confirm your selection and to 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.

---

TIP: Disabling deltarpms

By default updates are downloaded as deltarpms. Since rebuilding rpm packages from deltarpms is a memory and CPU time consuming task, certain setups or hard-

ware configurations might require you to disable the usage of deltarpms for performance sake.

To disable the use of deltarpms edit the file `/etc/zypp/zypp.conf` and set `download.use_deltarpm` to `false`.

## 6.3 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.

Procedure 6.2 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 whether to update Daily, Weekly, or Monthly.

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

4. To automatically accept any license agreements, activate Agree with Licenses.

5. Select if you also 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 from time to time in order to install those patches, too. Otherwise you might miss important patches.

6. Confirm your configuration with OK.

By default, it is only possible to install packages from configured and enabled repositories. Apart from the official repositories that are configured during the installation, numerous other repositories exist. The openSUSE® Build Service hosts several hundred ones and a lot of third party repositories exist, too—see http://en.opensuse.org/Additional_package_repositories.

openSUSE offers two easy ways to install from these repositories without the need to first subscribe to them. The 1-Click Install method allows you to install packages directly from a Web browser, while the YaST Package Search lets you query almost all known repositories for openSUSE. You can directly install any package found by the package search module.

---

WARNING: Trusting External Software Sources

Before installing from any external software repository, make sure it can be trusted. openSUSE is not responsible for any potential problems arising from software installed from third-party software repositories.

---

## 7.1 1-Click Install

Installation using 1-Click Install is available from a lot of repositories available on Web interfaces. A very popular interface is the openSUSE Build Service software search.

Procedure 7.1 Installing Packages from the openSUSE Build Service via 1-Click Install

1. Start the openSUSE Build Service search interface at http://software.opensuse.org/search.
2. Select your system version from the drop-down menu, for example openSUSE 12.2.
3. Enter the name of the package you want to install, for example the OpenStreetMap editor josm.
4. To refine the search, adjust the Search Options according to your wishes.
5. Click Search.
6. From the results list select the preferred item by clicking its 1-Click Install button.
7. In the Web browser's download dialog, select to open the file with the YaST Meta Package Handler.

The 1-click installer opens the Additional Software Repositories dialog. It shows the repositories providing the packages you want to install per `1-Click Install`. They are activated per default. To proceed with the installation, keep the repository selection. By default, you remain subscribed to these repositories after the installation has finished and you will receive updates from them in the future.

8 If you want to use the new repositories only once instead, uncheck Remain Subscribed to These Repositories after Installation. Click Next to proceed.

9 Now select the software packages that should be installed. Normally there is no need to change the default selection. Click Next to proceed.

10 The Proposal screen summarizes the choices you made. Click Customize to restart the configuration steps from above. Click Next and Yes to proceed with the installation.

11 Enter the `root` password to start the installation. In case a new repository has been added you also need to confirm the import of the repository's GPG key. During the installation several progress pop-ups appear that do not need any interaction. After reading the “Installation was successful” message, click Finish.

---

TIP: Disabling 1-Click Install Feature

If you want to disable the 1-Click install feature, uninstall the `yast2-metapackage-handler` package using YaST. Alternatively, enter the following command as `root`:

```
rpm -e yast2-metapackage-handler
```

---

## 7.2 YaST Package Search

Provided you are connected to the internet, you can also search and install packages from almost all known repositories for openSUSE directly via YaST's Package Search. This module is a YaST frontend for the Webpin package search available at http://packages.opensuse-community.org/. The YaST module is not available by default—you need to install the package `yast2-packager-webpin` first.

Procedure 7.2 Installing Packages Using the YaST Package Search

1 Start the Package Search from the YaST Control Center via Software > Package Search.

2 Search for a software package by entering it's name into the Search Expression field and clicking Search.

3 The search result is listed on the Found Packages tab. Click on a package name to see the repository URL, the package version number and the architecture in the Package Description pane.

WARNING: Double Check the Package Information

Make sure to double check whether the software is hosted on a repository you trust before you install it. Also check if the architecture complies with your system (x86_64 packages can only be installed on 64bit systems).

4 Mark a package for installation by activating its checkbox. You can mark several packages at once. You can even start a new search for other packages without losing your current selection, which is always available on the All Selected Packages. Once you have finished the package selection, proceed with Next.

The Additional Software Repositories dialog shows the repositories providing the packages you want to install. They are activated per default. To proceed with the installation, keep the repository selection. By default, you remain subscribed to these repositories after the installation has finished, so you will receive updates from them in the future.

5 If instead you want to use the new repositories only once, uncheck Remain Subscribed to These Repositories after Installation. Click Next to proceed.

6 Now select the software packages that should be installed. Normally there is no need to change the default selection. Click Next to proceed.

7 The Proposal screen summarizes the choices you made. Click Customize to restart the configuration steps from above. Click Next and Yes to proceed with the installation.

8 Confirm the next dialog. In case a new repository is used, you will also need to confirm the import of the repository's GPG key. During the installation several progress pop-ups appear that do not need any interaction. After reading the “Installation was successful” message, click Finish.

Add-on products are system extensions. You can install a third party add-on product or a special system extension of openSUSE® (for example, a CD with support for additional languages or a CD with binary drivers). To install a new add-on, start YaST and select Software > Add-On Products . You can select various types of product media, like CD, FTP, USB mass storage devices (such as USB flash drives or disks) or a local directory. You can work also directly with ISO files. To add an add-on as ISO file media, select Local ISO Image then enter the Path to ISO Image. The Repository Name is arbitrary.

## 8.1 Add-Ons

To install a new add-on, proceed as follows:

1. In YaST select Software > Add-On Products to see an overview of already installed add-on products.

2. To install a new add-on product, click Add.

3. From the list of available Media Types specify the type matching your repository.

4. To add a repository from a removable medium, choose the relevant option and insert the medium or connect the USB device to the machine, respectively.

5. You can choose to Download Repository Description Files now. If the option is unchecked, YaST will automatically download the files later, if needed. Click Next to proceed.

6. When adding a repository from the network, enter the data you are prompted for. Continue with Next.

7. Depending on the repository you have added, you may be asked if you want to import the GPG key with which it is signed or asked to agree to a license.

   After confirming these messages, YaST will download and parse the metadata and add the repository to the list of Configured Repositories..

8. If needed, adjust the repository Properties as described in Section 5.4.2, “Managing Repository Properties” (page 90) or confirm your changes with OK to close the configuration dialog.

9. After having successfully added the repository for the add-on media, the software manager starts and you can install packages. Refer to Chapter 5, Installing or Removing Software (page 77) for details.

## 8.2 Binary Drivers

Some hardware needs binary-only drivers to function properly. If you have such hardware, refer to the release notes for more information about availability of binary drivers for your system. To read the release notes, open YaST and select Miscellaneous > Release Notes.

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 Section 5.1, “Definition of Terms” (page 77).

# 9.1 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.

## 9.1.1 General Usage

The general syntax of Zypper is:

```
zypper [global-options]command[command-options][arguments] ...
```

The components enclosed in brackets are not required. 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 type:

```
zypper patch
```

Additionally, you can choose from one or more global options by typing them just before the command. For example, `--non-interactive` means running the command without asking anything (automatically applying the default answers):

```
zypper --non-interactive patch
```

To use the options specific to a particular command, type them right after the command. For example, `--auto-agree-with-licenses` means applying all needed patches to the system without asking to confirm any licenses (they will automatically be accepted):

```
zypper patch --auto-agree-with-licenses
```

Some commands require one or more arguments. When using the install command, for example, you need to specify which package(s) to install:

```
zypper install mplayer
```

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

```
zypper search -t pattern
```

You can combine all of the above. For example, the following command will install the `mplayer` and `amarok` packages from the `factory` repository while being verbose:

```
zypper -v install --from factory mplayer amarok
```

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.

```
zypper remove --dry-run MozillaFirefox
```

### 9.1.2 Installing and Removing Software with Zypper

To install or remove packages use the following commands:

```
zypper install package_name
zypper remove package_name
```

Zypper knows various ways to address packages for the install and remove commands:

by the exact package name (and version number)

```
zypper install MozillaFirefox
```

or

```
zypper install MozillaFirefox-3.5.3
```

by repository alias and package name

```
zypper install mozilla:MozillaFirefox
```

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

by package name using wildcards

The following command will install all packages that have names starting with “Moz”. Use with care, especially when removing packages.

```
zypper install 'Moz*'
```

by capability

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

```
zypper install 'perl(Time::ParseDate)'
```

by capability and/or architecture and/or version

Together with a capability you can specify an architecture (such as `i586` or `x86_64`) and/or a version. The version must be preceded by an operator: `<` (lesser than), `<=` (lesser than or equal), `=` (equal), `>=` (greater than or equal), `>` (greater than).

```
zypper install 'firefox.x86_64'
zypper install 'firefox>=3.5.3'
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:

```
zypper install /tmp/install/MozillaFirefox.rpm
zypper install http://download.opensuse.org/
repositories/mozilla/SUSE_Factory/x86_64/
MozillaFirefox-3.5.3-1.3.x86_64.rpm
```

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

```
zypper install emacs -vim
```

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

```
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 `--`:

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

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:

```
rm package_name --clean-deps
```

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 in the following:

```
zypper --non-interactive install package_name
```

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

---

WARNING: Do not Remove Mandatory System Packages

Do not remove packages such as `glibc`, `zypper`, `kernel`, or similar packages. These packages are mandatory for the system and, if removed, may cause the system to become unstable or stop working altogether.

---

### 9.1.2.1 Installing Source Packages

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

```
zypper source-install package_name
```

That 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`.

```
zypper source-install -D package_name # source package only
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 9.1.4, “Managing Repositories with Zypper” (page 115) for details on repository management.

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

```
zypper search -t srcpackage
```

#### 9.1.2.2 Utilities

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

```
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:

```
zypper install-new-recommends
```

This command is very useful after plugging in a webcam or WLAN 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.

### 9.1.3 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 the `zypper dist-upgrade` command which is discussed in Section 16.1, “Upgrading the System” (page 177).

#### 9.1.3.1 Installing Patches

To install all officially released patches applying to your system, just run:

```
zypper patch
```

In this case, all patches available in your repositories are checked for relevance and installed, if necessary. The above command is all you must enter in order to apply them when needed.

Zypper knows three different commands to query for the availability of patches:

`zypper patch-check`
: Lists the number of needed patches (patches, that apply to your system but are not yet installed)

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

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

```
~ # zypper list-patches
Loading repository data...
Reading installed packages...

Repository                          | Name      | Version | Category | Status
------------------------------------+-----------+---------+----------+-------
Updates for openSUSE 11.3 11.3-1.82 | lxsession | 2776    | security | needed
```

`zypper patches`
: Lists all patches available for openSUSE, regardless of whether they are already installed or apply to your installation.

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:

`--bugzilla[=`*number*`]`
: Lists all needed patches for Bugzilla issues. Optionally, you can specify a bug number if you only want to list patches for this specific bug.

`--cve[=`*number*`]`
: Lists all needed patches for CVE (Common Vulnerabilities and Exposures) issues, or only patches matching a certain CVE number, if specified.

To install a patch for a specific Bugzilla or CVE issue, use the following commands:

`zypper patch --bugzilla=`*number*

or

`zypper patch --cve=`*number*

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

`zypper patch --cve=CVE-2010-2713`

### 9.1.3.2 Installing Updates

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, use:

`zypper update`

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

```
zypper update package_name
zypper install package_name
```

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

```
zypper list-updates
```

Note that this command only packages 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:

```
zypper list-updates --all
```

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

### 9.1.3.3 Upgrading to a New Product Version

To easily upgrade your installation to a new product version (for example, from openSUSE 11.4 to openSUSE 12.1), first adjust your repositories to match the current openSUSE repositories. For details, refer to Section 9.1.4, “Managing Repositories with Zypper” (page 115). Then use the `zypper dist-upgrade` command with the required repositories. This command ensures that all packages will be installed from the repositories currently enabled. For detailed instructions, refer to Section 16.1.4, “Distribution Upgrade with zypper” (page 179).

To restrict the distribution upgrade to packages from a certain repository while considering also the other repositories for satisfying dependencies, use the `--from` option and specify the repository by either its alias, its number or URI.

---

NOTE: Differences between `zypper update` and `zypper dist-upgrade`

Choose `zypper update` to update packages to newer versions available for your product version while maintaining system integrity. `zypper update` will honor the following rules:

no vendor changes
no architecture changes
no downgrades
keep installed packages

When executing `zypper dist-upgrade`, all packages will be installed from the repositories currently enabled. This rule is enforced, so packages might change

vendor or architecture or even might get downgraded. All packages that have unfulfilled dependencies after the upgrade will be uninstalled.

## 9.1.4 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:

```
zypper repos
```

The result will look similar to the following output:

Example 9.1 Zypper—List of Known Repositories

```
# | Alias                 | Name                  | Enabled | Refresh
--+-----------------------+-----------------------+---------+--------
1 | Updates               | Updates               | Yes     | Yes
2 | openSUSE 11.2-0       | openSUSE 11.2-0       | No      | No
3 | openSUSE-11.2-Debug   | openSUSE-11.2-Debug   | No      | Yes
4 | openSUSE-11.2-Non-Oss | openSUSE-11.2-Non-Oss | Yes     | Yes
5 | openSUSE-11.2-Oss     | openSUSE-11.2-Oss     | Yes     | Yes
6 | openSUSE-11.2-Source  | openSUSE-11.2-Source  | No      | Yes
```

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:

```
zypper repos -d
```

### 9.1.4.1 Adding Repositories

To add a repository, run

```
zypper addrepo URIalias
```

*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 is has to be unique. Zypper will issue a warning if you specify an alias that is already in use.

### 9.1.4.2 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 listed as third entry in Example 9.1, “Zypper—List of Known Repositories” (page 115), use the following command:

```
zypper removerepo 3
```

#### 9.1.4.3 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:

```
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
`-t`: 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 just `firefox`:

```
zypper renamerepo 'Mozilla Firefox' firefox
```

### 9.1.5 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:

```
zypper products
zypper patterns
zypper packages
zypper patches
```

To query all repositories for certain packages, use `search`. It works on package names, or, optionally, on package summaries and descriptions. Using the wildcards `*` and `?` with the search term is allowed. By default, the search is not case-sensitive.

```
zypper search firefox       # simple search for "firefox"
zypper search "*fire*"      # using wildcards
zypper search -d fire       # also search in package
 descriptions and summaries
zypper search -u firefox    # only display packages not
 already installed
```

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

```
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`:

```
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.

### 9.1.6 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 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.

### 9.1.7 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:

```
zypper refresh
```

If that does not help, try

```
zypper refresh -fdb
```

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

### 9.1.8 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. For more information about `snapper`, see `man snapper`.

Zypper (and YaST) currently only make snapshots of the root filesystem. Other subvolumes cannot be configured. This feature is not supported on the default file system.

### 9.1.9 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)` manpage. For a com-

plete 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 openSUSE version can be found at . [`http://en.opensuse.org/openSUSE:Zypper versions`] .

## 9.2 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 a number of 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`, `gimp-devel`, and `libkde4-devel`.

---

### 9.2.1 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 Novell/SUSE or from another trustworthy facility. This is especially recommended for update packages from the Internet.

### 9.2.2 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 as 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.rpm`, which 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`.

## 9.2.3 RPM and Patches

To guarantee the operational security of a system, update packages must be installed in the system from time to time. Previously, a bug in a package could only be eliminated by replacing the entire package. Large packages with bugs in small files could easily result in this scenario. However the SUSE RPM offers a feature enabling the installation of patches in packages.

The most important considerations are demonstrated using `pine` as an example:

Is the patch RPM suitable for my system?
: To check this, first query the installed version of the package. For `pine`, this can be done with

```
rpm -q pine
pine-4.44-188
```

Then check if the patch RPM is suitable for this version of `pine`:

```
rpm -qp --basedon pine-4.44-224.i586.patch.rpm
pine = 4.44-188
pine = 4.44-195
pine = 4.44-207
```

This patch is suitable for three different versions of pine. The installed version in the example is also listed, so the patch can be installed.

Which files are replaced by the patch?
: The files affected by a patch can easily be seen in the patch RPM. The `rpm` parameter `-P` allows selection of special patch features. Display the list of files with the following command:

```
rpm -qpPl pine-4.44-224.i586.patch.rpm
/etc/pine.conf
/etc/pine.conf.fixed
/usr/bin/pine
```

or, if the patch is already installed, with the following command:

```
rpm -qPl pine
/etc/pine.conf
/etc/pine.conf.fixed
/usr/bin/pine
```

How can a patch RPM be installed in the system?
: Patch RPMs are used just like normal RPMs. The only difference is that a suitable RPM must already be installed.

Which patches are already installed in the system and for which package versions?
: A list of all patches installed in the system can be displayed with the command `rpm -qPa`. If only one patch is installed in a new system (as in this example), the list appears as follows:

```
rpm -qPa
pine-4.44-224
```

If, at a later date, you want to know which package version was originally installed, this information is also available in the RPM database. For `pine`, this information can be displayed with the following command:

```
rpm -q --basedon pine
```

```
pine = 4.44-188
```

More information, including information about the patch feature of RPM, is available in the man pages of `rpm` and `rpmbuild`.

---

NOTE: Official Updates for openSUSE

In order to make the download size of updates as small as possible, official updates for openSUSE are not provided as Patch RPMs, but as Delta RPM packages. For details, see Section 9.2.4, “Delta RPM Packages” (page 121).

---

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

```
prepdeltarpm -s seq -i info old.rpm > old.cpio
prepdeltarpm -f new.rpm > new.cpio
xdelta delta -0 old.cpio new.cpio delta
writedeltarpm new.rpm delta info new.delta.rpm
```

Finally, remove the temporary working files `old.cpio`, `new.cpio`, and `delta`.

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.

## 9.2.5 RPM Queries

With the `-q` option `rpm` initiates queries, making it possible to inspect an RPM archive (by adding the option `-p`) and also to query the RPM database of installed packages. Several switches are available to specify the type of information required. See Table 9.1, “The Most Important RPM Query Options” (page 122).

Table 9.1 The Most Important RPM Query Options

| | |
|---|---|
| `-i` | Package information |
| `-l` | File list |
| `-f FILE` | Query the package that contains the file *`FILE`* (the full path must be specified with *`FILE`*) |
| `-s` | File list with status information (implies `-l`) |
| `-d` | List only documentation files (implies `-l`) |
| `-c` | List only configuration files (implies `-l`) |
| `--dump` | File list with complete details (to be used with `-l`, `-c`, or `-d`) |
| `--provides` | List features of the package that another package can request with `--requires` |
| `--requires`, `-R` | Capabilities the package requires |
| `--scripts` | Installation scripts (preinstall, postinstall, uninstall) |

For example, the command `rpm -q -i wget` displays the information shown in Example 9.2, “rpm -q -i wget” (page 122).

Example 9.2 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 filename with its full path. Provide as many filenames 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 filename is known, use a shell script as shown in Example 9.3, “Script to Search for Packages” (page 123). Pass the partial filename to the script shown as a parameter when running it.

Example 9.3 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 rpm` displays a detailed list of change information about a specific package (in this case, the `rpm`package), sorted by date.

With the help of 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 9.2 RPM Verify Options

| | |
|---|---|
| `5` | MD5 check sum |
| `S` | File size |
| `L` | Symbolic link |

| | |
|---|---|
| T | Modification time |
| D | Major and minor device numbers |
| U | Owner |
| G | Group |
| M | 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_RPMDB_BACKUPS` (default: `5`) in `/etc/sysconfig/backup`. The size of a single backup is approximately 1 MB for 1 GB in `/usr`.

## 9.2.6 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

Do not experiment with system components (`glibc`, `rpm`, `sysvinit`, 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.

### 9.2.7 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 a number of 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.

### 9.2.8 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 Chapter 5, Installing or Removing Software (page 77).

# Part III. Administration

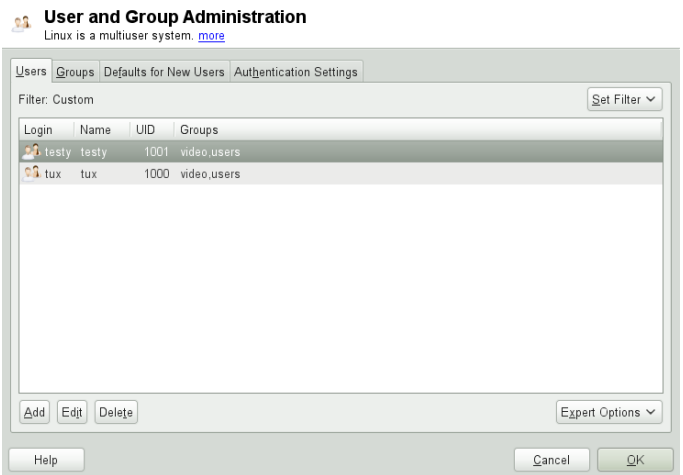
During installation, you chose a method for user authentication. This method is either local (via `/etc/passwd`) or, if a network connection is established, via NIS, LDAP, Kerberos or Samba (see Section “Create New User” (Chapter 1, Installation with YaST, ↑Reference) . You can create or modify user accounts and change the authentication method with YaST at any time.

Every user is assigned a system-wide user ID (UID). Apart from the users which can log in to your machine, there are also a number of system users for internal use only. Each user is assigned to one or more groups. Similar to system users, there are also system groups for internal use.

## 10.1 User and Group Administration Dialog

To administer users or groups, start YaST and click Security and Users > User and Group Management. Alternatively, start the User and Group Administration dialog directly by running `yast2 users &` from a command line.

Figure 10.1 YaST User and Group Administration




Depending on the set of users you choose to view and modify with, the dialog (local users, network users, system users), the main window shows several tabs. These allow you to execute the following tasks:

Managing User Accounts
: From the Users tab create, modify, delete or temporarily disable user accounts as described in Section 10.2, “Managing User Accounts” (page 130). Learn about advanced options like enforcing password policies, using encrypted home directories, using fingerprint authentication, or managing disk quotas in Section 10.3, “Additional Options for User Accounts” (page 132).

Changing Default Settings
: Local users accounts are created according to the settings defined on the Defaults for New Users tab. Learn how to change the default group assignment, or the default path and access permissions for home directories in Section 10.4, “Changing Default Settings for Local Users” (page 137).

Assigning Users to Groups
: Learn how to change the group assignment for individual users in Section 10.5, “Assigning Users to Groups” (page 137).

Managing Groups
: From the Groups tab, you can add, modify or delete existing groups. Refer to Section 10.6, “Managing Groups” (page 138) for information on how to do this.

Changing the User Authentication Method
: When your machine is connected to a network that provides user authentication methods like NIS or LDAP, you can choose between several authentication methods on the Authentication Settings tab. For more information, refer to Section 10.7, “Changing the User Authentication Method” (page 139).

For user and group management, the dialog provides similar functionality. You can easily switch between the user and group administration view by choosing the appropriate tab at the top of the dialog.

Filter options allow you to define the set of users or groups you want to modify: On the Users or Group tab, click Set Filter to view and edit users or groups according to certain categories, such as Local Users or LDAP Users, for instance (if you are part of a network which uses LDAP). With Set Filter > Customize Filter you can also set up and use a custom filter.

Depending on the filter you choose, not all of the following options and functions will be available from the dialog.

## 10.2 Managing User Accounts

YaST offers to create, modify, delete or temporarily disable user accounts. Do not modify user accounts unless you are an experienced user or administrator.

---

NOTE: Changing User IDs of Existing Users

File ownership is bound to the user ID, not to the user name. After a user ID change, the files in the user's home directory are automatically adjusted to reflect this change. However, after an ID change, the user no longer owns the files he

created elsewhere in the file system unless the file ownership for those files are manually modified.

---

In the following, learn how to set up default user accounts. For some further options, such as auto login, login without password, setting up encrypted home directories or managing quotas for users and groups, refer to Section 10.3, “Additional Options for User Accounts” (page 132).

Procedure 10.1 Adding or Modifying User Accounts

1. Open the YaST User and Group Administration dialog and click the Users tab.
2. With Set Filter define the set of users you want to manage. The dialog shows a list of users in the system and the groups the users belong to.
3. To modify options for an existing user, select an entry and click Edit.

   To create a new user account, click Add.
4. Enter the appropriate user data on the first tab, such as Username (which is used for login) and Password. This data is sufficient to create a new user. If you click OK now, the system will automatically assign a user ID and set all other values according to the default.
5. Activate Receive System Mail if you want any kind of system notifications to be delivered to this user's mailbox. This creates a mail alias for `root` and the user can read the system mail without having to first log in as `root`.
6. If you want to adjust further details such as the user ID or the path to the user's home directory, do so on the Details tab.

   If you need to relocate the home directory of an existing user, enter the path to the new home directory there and move the contents of the current home directory with Move to New Location. Otherwise, a new home directory is created without any of the existing data.
7. To force users to regularly change their password or set other password options, switch to Password Settings and adjust the options. For more details, refer to Section 10.3.2, “Enforcing Password Policies” (page 132).
8. If all options are set according to your wishes, click OK.
9. Click Expert Options > Write Changes Now to save all changes without exiting the User and Group Administration dialog. Click OK to close the administration dialog and to save the changes. A newly added user can now log in to the system using the login name and password you created.

---

TIP: Matching User IDs

For a new (local) user on a laptop which also needs to integrate into a network environment where this user already has a user ID, it is useful to match the (local)

user ID to the ID in the network. This ensures that the file ownership of the files the user creates “offline” is the same as if he had created them directly on the network.

Procedure 10.2 Disabling or Deleting User Accounts

1. Open the YaST User and Group Administration dialog and click the Users tab.
2. To temporarily disable a user account without deleting it, select the user from the list and click Edit. Activate Disable User Login. The user cannot log into your machine until you enable the account again.
3. To delete a user account, select the user from the list and click Delete. Choose if you also want to delete the user's home directory or if you want to retain the data.

## 10.3 Additional Options for User Accounts

In addition to the settings for a default user account, openSUSE® offers further options, such as options to enforce password policies, use encrypted home directories or define disk quotas for users and groups.

### 10.3.1 Automatic Login and Passwordless Login

If you use the KDE or GNOME desktop environment you can configure Auto Login for a certain user as well as Passwordless Login for all users. Auto login causes a user to become automatically logged in to the desktop environment on boot. This functionality can only be activated for one user at a time. Login without password allows all users to log in to the system after they have entered their username in the login manager.

WARNING: Security Risk

Enabling Auto Login or Passwordless Login on a machine that can be accessed by more than one person is a security risk. Without the need to authenticate, any user can gain access to your system and your data. If your system contains confidential data, do not use this functionality.

If you want to activate auto login or login without password, access these functions in the YaST User and Group Administration with Expert Options > Login Settings.

### 10.3.2 Enforcing Password Policies

On any system with multiple users, it is a good idea to enforce at least basic password security policies. Users should change their passwords regularly and use strong passwords that cannot easily be exploited. For local users, proceed as follows:

Procedure 10.3 Configuring Password Settings

1. Open the YaST User and Group Administration dialog and select the Users tab.

2 Select the user for which to change the password options and click Edit.

3 Switch to the Password Settings tab. The user's last password change is displayed on the tab.

4 To make the user change his password at next login, activate Force Password Change.

5 To enforce password rotation, set a Maximum Number of Days for the Same Password and a Minimum Number of Days for the Same Password.

6 To remind the user to change his password before it expires, set a number of Days before Password Expiration to Issue Warning.

7 To restrict the period of time the user can log in after his password has expired, change the value in Days after Password Expires with Usable Login.

8 You can also specify a certain expiration date for a password. Enter the Expiration Date in *YYYY-MM-DD* format.

9 For more information about the options and about the default values, click Help.

10 Apply your changes with OK.

### 10.3.3 Managing Encrypted Home Directories

To protect data in home directories against theft and hard disk removal, you can create encrypted home directories for users. These are encrypted with LUKS (Linux Unified Key Setup), which results in an image and an image key being generated for the user. The image key is protected with the user's login password. When the user logs into the system, the encrypted home directory is mounted and the contents are made available to the user.

---

NOTE: Fingerprint Reader Devices and Encrypted Home Directories

If you want to use a fingerprint reader device, you must not use encrypted home directories. Otherwise logging in will fail, because decrypting during login is not possible in combination with an active fingerprint reader device.

---

With YaST, you can create encrypted home directories for new or existing users. To encrypt or modify encrypted home directories of already existing users, you need to know the user's current login password. By default, all existing user data is copied to the new encrypted home directory, but it is not deleted from the unencrypted directory.

---

WARNING: Security Restrictions

Encrypting a user's home directory does not provide strong security from other users. If strong security is required, the system should not be physically shared.

---

Find background information about encrypted home directories and which actions to take for stronger security in Section “Using Encrypted Home Directories” (Chapter 10, Encrypting Partitions and Files, ↑Security Guide).

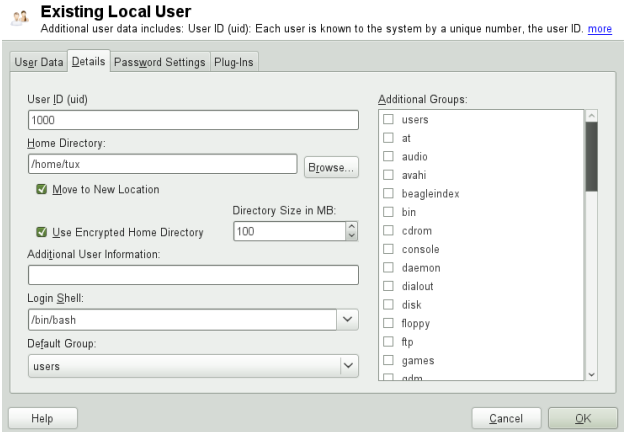
Procedure 10.4 Creating Encrypted Home Directories

1 Open the YaST User and Group Management dialog and click the Users tab.

2 To encrypt the home directory of an existing user, select the user and click Edit.

Otherwise, click Add to create a new user account and enter the appropriate user data on the first tab.

3 In the Details tab, activate Use Encrypted Home Directory. With Directory Size in MB, specify the size of the encrypted image file to be created for this user.




4 Apply your settings with OK.

5 Enter the user's current login password to proceed if YaST prompts for it.

6 Click Expert Options > Write Changes Now to save all changes without exiting the administration dialog. Click OK to close the administration dialog and save the changes.

Procedure 10.5 Modifying or Disabling Encrypted Home Directories

Of course, you can also disable the encryption of a home directory or change the size of the image file at any time.

1 Open the YaST User and Group Administration dialog in the Users view.

2 Select a user from the list and click Edit.

3 If you want to disable the encryption, switch to the Details tab and disable Use Encrypted Home Directory.

   If you need to enlarge or reduce the size of the encrypted image file for this user, change the Directory Size in MB.

4 Apply your settings with OK.

5 Enter the user's current login password to proceed if YaST prompts for it.

6 Click Expert Options > Write Changes Now to save all changes without exiting the User and Group Administration dialog. Click OK to close the administration dialog and to save the changes.

### 10.3.4 Using Fingerprint Authentication

If your system includes a fingerprint reader you can use biometric authentication in addition to standard authentication via login and password. After registering their fingerprint, users can log into the system either by swiping a finger on the fingerprint reader or by typing in a password.

Fingerprints can be registered with YaST. Find detailed information about configuration and use of fingerprint authentication in Chapter 7, Using the Fingerprint Reader (↑Security Guide). For a list of supported devices, refer to http://www.freedesktop.org/wiki/Software/fprint/libfprint.

### 10.3.5 Managing Quotas

To prevent system capacities from being exhausted without notification, system administrators can set up quotas for users or groups. Quotas can be defined for one or more file systems and restrict the amount of disk space that can be used and the number of inodes (index nodes) that can be created there. Inodes are data structures on a file system that store basic information about a regular file, directory, or other file system object. They store all attributes of a file system object (like user and group ownership, read, write, or execute permissions), except file name and contents.

openSUSE allows usage of `soft` and `hard` quotas. Soft quotas usually define a warning level at which users are informed that they are nearing their limit, whereas hard quotas define the limit at which write requests are denied. Additionally, grace intervals can be defined that allow users or groups to temporarily violate their quotas by certain amounts.

Procedure 10.6 Enabling Quota Support for a Partition

In order to configure quotas for certain users and groups, you need to enable quota support for the respective partition in the YaST Expert Partitioner first.

1 In YaST, select System > Partitioner and click Yes to proceed.

2 In the Expert Partitioner, select the partition for which to enable quotas and click Edit.

3 Click Fstab Options and activate Enable Quota Support. If the `quota` package is not already installed, it will be installed once you confirm the respective message with Yes.

4 Confirm your changes and leave the Expert Partitioner.

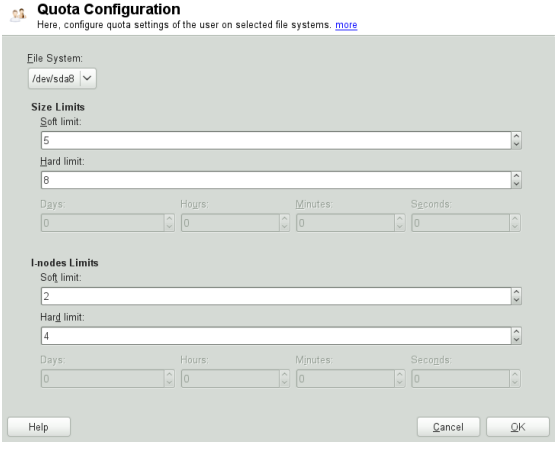
Procedure 10.7 Setting Up Quotas for Users or Groups

Now you can define soft or hard quotas for specific users or groups and set time periods as grace intervals.

1 In the YaST User and Group Administration, select the user or the group you want to set the quotas for and click Edit.

2 On the Plug-Ins tab, select the Manage User Quota entry and click Launch to open the Quota Configuration dialog.

3 From File System, select the partition to which the quota should apply.




4 Below Size Limits, restrict the amount of disk space. Enter the number of 1 KB blocks the user or group may have on this partition. Specify a Soft Limit and a Hard Limit value.

5 Additionally, you can restrict the number of inodes the user or group may have on the partition. Below Inodes Limits, enter a Soft Limit and Hard Limit.

6 You can only define grace intervals if the user or group has already exceeded the soft limit specified for size or inodes. Otherwise, the time-related input fields are not activated. Specify the time period for which the user or group is allowed to exceed the limits set above.

7 Confirm your settings with OK.

8 Click Expert Options > Write Changes Now to save all changes without exiting the User and Group Administration dialog. Click OK to close the administration dialog and to save the changes.

openSUSE also ships command line tools like `repquota` or `warnquota` with which system administrators can control the disk usage or send e-mail notifications to users exceeding their quota. With `quota_nld`, administrators can also forward kernel messages about exceeded quotas to D-BUS. For more information, refer to the `repquota`, the `warnquota` and the `quota_nld` man page.

## 10.4 Changing Default Settings for Local Users

When creating new local users, several default settings are used by YaST. These include, for example, the primary group and the secondary groups the user belongs to, or the access permissions of the user's home directory. You can change these default settings to meet your requirements:

1 Open the YaST User and Group Administration dialog and select the Defaults for New Users tab.

2 To change the primary group the new users should automatically belong to, select another group from Default Group.

3 To modify the secondary groups for new users, add or change groups in Secondary Groups. The group names must be separated by commas.

4 If you do not want to use `/home/`*`username`* as default path for new users' home directories, modify the Path Prefix for Home Directory.

5 To change the default permission modes for newly created home directories, adjust the umask value in Umask for Home Directory. For more information about umask, refer to Chapter 9, Access Control Lists in Linux (↑Security Guide) and to the `umask` man page.

6 For information about the individual options, click Help.

7 Apply your changes with OK.

## 10.5 Assigning Users to Groups

Local users are assigned to several groups according to the default settings which you can access from the User and Group Administration dialog on the Defaults for New Users tab. In the following, learn how to modify an individual user's group assignment. If you need to change the default group assignments for new users, refer to Section 10.4, “Changing Default Settings for Local Users” (page 137).

Procedure 10.8 Changing a User's Group Assignment

1 Open the YaST User and Group Administration dialog and click the Users tab. It shows a list of users and of the groups the users belong to.

2 Click Edit and switch to the Details tab.

3 To change the primary group the user belongs to, click Default Group and select the group from the list.

4 To assign the user additional secondary groups, activate the corresponding check boxes in the Additional Groups list.

5 Click OK to apply your changes.

6 Click Expert Options > Write Changes Now to save all changes without exiting the User and Group Administration dialog. Click OK to close the administration dialog and save the changes.

## 10.6 Managing Groups

With YaST you can also easily add, modify or delete groups.

Procedure 10.9 Creating and Modifying Groups

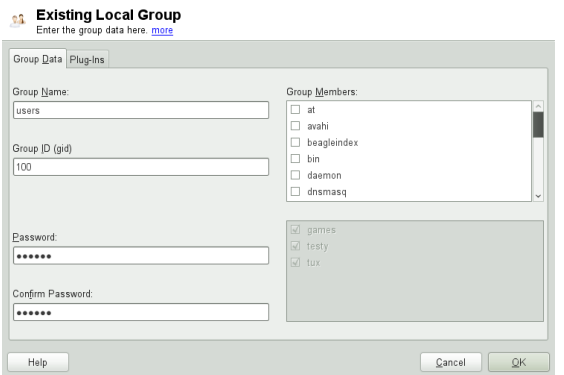
1 Open the YaST User and Group Management dialog and click the Groups tab.

2 With Set Filter define the set of groups you want to manage. The dialog shows a list of groups in the system.

3 To create a new group, click Add.

4 To modify an existing group, select the group and click Edit.

5 In the following dialog, enter or change the data. The list on the right shows an overview of all available users and system users which can be members of the group.



6 To add existing users to a new group select them from the list of possible Group Members by checking the corresponding box. To remove them from the group uncheck the box.

7 Click OK to apply your changes.

8 Click Expert Options > Write Changes Now to save all changes without exiting the User and Group Administration dialog.

In order to delete a group, it must not contain any group members. To delete a group, select it from the list and click Delete. Click Expert Options > Write Changes Now to save all changes without exiting the User and Group Administration dialog. Click OK to close the administration dialog and to save the changes.

## 10.7 Changing the User Authentication Method

When your machine is connected to a network, you can change the authentication method you set during installation. The following options are available:

NIS
: Users are administered centrally on a NIS server for all systems in the network. For details, see Chapter 3, Using NIS (↑Security Guide).

LDAP
: Users are administered centrally on an LDAP server for all systems in the network. For details about LDAP, see Chapter 4, LDAP—A Directory Service (↑Security Guide).

  You can manage LDAP users with the YaST user module. All other LDAP settings, including the default settings for LDAP users, have to be defined with the YaST LDAP client module as described in Section “Configuring an LDAP Client with YaST” (Chapter 4, LDAP—A Directory Service, ↑Security Guide) .

Kerberos
: With Kerberos, a user registers once and then is trusted in the entire network for the rest of the session.

Samba
: SMB authentication is often used in mixed Linux and Windows networks. For details, see Chapter 15, Samba (↑Reference) and Chapter 5, Active Directory Support (↑Security Guide).

To change the authentication method, proceed as follows:

1 Open the User and Group Administration dialog in YaST.

2 Click the Authentication Settings tab to show an overview of the available authentication methods and the current settings.

3 To change the authentication method, click Configure and select the authentication method you want to modify. This takes you directly to the client config-

uration modules in YaST. For information about the configuration of the appropriate client, refer to the following sections:

NIS: Section “Configuring NIS Clients” (Chapter 3, Using NIS, ↑Security Guide)

LDAP: Section “Configuring an LDAP Client with YaST” (Chapter 4, LDAP—A Directory Service, ↑Security Guide)

4 After accepting the configuration, return to the User and Group Administration overview.

5 Click OK to close the administration dialog.

Working in different countries or having to work in a multilingual environment requires your computer to be set up to support this. openSUSE® can handle different `locales` in parallel. A locale is a set of parameters that defines the language and country settings reflected in the user interface.

The main system language was selected during installation and keyboard and time zone settings were adjusted. However, you can install additional languages on your system and determine which of the installed languages should be the default.

For those tasks, use the YaST language module as described in Section 11.1, “Changing the System Language” (page 141). Install secondary languages to get optional localizations if you need to start applications or desktops in languages other than the primary one.

Apart from that, the YaST timezone module allows you to adjust your country and timezone settings accordingly. It also lets you synchronize your system clock against a time server. For details, refer to Section 11.2, “Changing the Country and Time Settings” (page 144).

## 11.1 Changing the System Language

Depending on how you use your desktop and whether you want to switch the entire system to another language or just the desktop environment itself, there are several ways to achieve this:

Changing the System Language Globally
: Proceed as described in Section 11.1.1, “Modifying System Languages with YaST” (page 142) and Section 11.1.2, “Switching the Default System Language” (page 143) to install additional localized packages with YaST and to set the default language. Changes are effective after relogin. To ensure that the entire system reflects the change, reboot the system or close and restart all running services, applications, and programs.

Changing the Language for the Desktop Only
: Provided you have previously installed the desired language packages for your desktop environment with YaST as described below, you can switch the language of your desktop using the desktop's control center. After the X server has been restarted, your entire desktop reflects your new choice of language. Applications not belonging to your desktop framework are not affected by this change and may still appear in the language that was set in YaST.

Temporarily Switching Languages for One Application Only
: You can also run a single application in another language (that has already been installed with YaST). To do so, start it from the command line by specifying the

language code as described in Section 11.1.3, “Switching Languages for Individual Applications” (page 144).

## 11.1.1 Modifying System Languages with YaST

YaST knows two different language categories:

Primary Language
: The primary language set in YaST applies to the entire system, including YaST and the desktop environment. This language is used whenever available unless you manually specify another language.

Secondary Languages
: Install secondary languages to make your system multilingual. Languages installed as secondary languages can be selected manually for a specific situation. For example, use a secondary language to start an application in a certain language in order to do word processing in this language.

Before installing additional languages, determine which of them should be the default system language (primary language) after you have installed them.

To access the YaST language module, start YaST and click System > Language. Alternatively, start the Languages dialog directly by running `yast2 language &` as user `root` from a command line.

Procedure 11.1 Installing Additional Languages

When installing additional languages, YaST also allows you to set different locale settings for the user `root`, see Step 4 (page 143). The option Locale Settings

for User root determines how the locale variables (`LC_*`) in the file `/etc/sysconfig/language` are set for `root`. You can either set them to the same locale as for normal users, keep it unaffected by any language changes or only set the variable `RC_LC_CTYPE` to the same values as for the normal users. This variable sets the localization for language-specific function calls.

1. To add additional languages in the YaST language module, select the Secondary Languages you wish to install.
2. To make a language the default language, set it as Primary Language.
3. Additionally, adapt the keyboard to the new primary language and adjust the time zone, if appropriate.

   ---

   TIP

   For advanced keyboard or time zone settings, select Hardware > Keyboard Layout or System > Date and Time in YaST to start the respective dialogs. For more information, refer to Section 11.2, “Changing the Country and Time Settings” (page 144).

   ---

4. To change language settings specific to the user `root`, click Details.

   4a Set Locale Settings for User root to the desired value. For more information, click Help.

   4b Decide if you want to Use UTF-8 Encoding for `root` or not.

5. If your locale was not included in the list of primary languages available, try specifying it with Detailed Locale Setting. However, some of these localizations may be incomplete.
6. Confirm your changes in the dialogs with OK. If you have selected secondary languages, YaST installs the localized software packages for the additional languages.

The system is now multilingual. However, to start an application in a language other than the primary one, you need to set the desired language explicitly as explained in Section 11.1.3, “Switching Languages for Individual Applications” (page 144).

### 11.1.2 Switching the Default System Language

1. To globally switch the default system language, start the YaST language module.
2. Select the desired new system language as Primary Language.

   ---

   IMPORTANT: Deleting Former System Languages

   If you switch to a different primary language, the localized software packages for the former primary language will be removed from the system. If you want

to switch the default system language but want to keep the former primary language as additional language, add it as Secondary Language by enabling the respective checkbox.

3 Adjust the keyboard and time zone options as desired.

4 Confirm your changes with OK.

5 After YaST has applied the changes, restart any X sessions (for example, by logging out and logging in again) to make YaST and the desktop applications reflect your new language settings.

### 11.1.3 Switching Languages for Individual Applications

After you have installed the respective language with YaST, you can run a single application in another language.

Standard X and GNOME Applications
: Start the application from the command line by using the following command:

  `LANG=`*`languageapplication`*

  For example, to start f-spot in German, run `LANG=de_DE f-spot`. For other languages, use the appropriate language code. Get a list of all language codes available with the `locale -av` command.

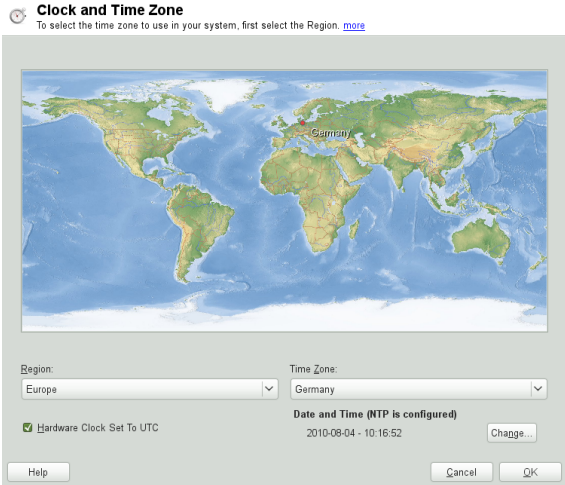
KDE Applications
: Start the application from the command line by using the following command:

  `KDE_LANG=`*`languageapplication`*

  For example, to start digiKam in German, run `KDE_LANG=de digikam`. For other languages, use the appropriate language code.

## 11.2 Changing the Country and Time Settings

Using the YaST date and time module, adjust your system date, clock and time zone information to the area you are working in. To access the YaST module, start YaST and click System > Date and Time. Alternatively, start the Clock and Time Zone dialog directly by running `yast2 timezone &` as user `root` from a command line.




First, select a general region, such as Europe. Choose an appropriate country that matches the one you are working in, for example, Germany.

Depending on which operating systems run on your workstation, adjust the hardware clock settings accordingly:

- If you run another operating system on your machine, such as Microsoft Windows*, it is likely your system does not use UTC, but local time. In this case, uncheck Hardware Clock Set To UTC.
- If you only run Linux on your machine, set the hardware clock to UTC and have the switch from standard time to daylight saving time performed automatically.

You can change the date and time manually or opt for synchronizing your machine against an NTP server, either permanently or just for adjusting your hardware clock.

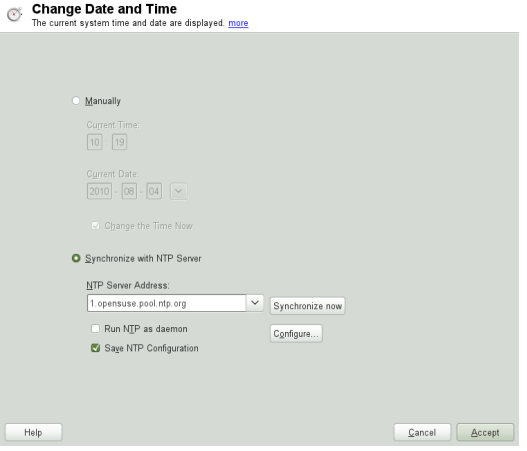
Procedure 11.2 Manually Adjusting Time and Date

1. In the YaST timezone module, click Change to set date and time.
2. Select Manually and enter date and time values.
3. Confirm your changes with Accept.

Procedure 11.3 Setting Date and Time With NTP Server

1. Click Change to set date and time.
2. Select Synchronize with NTP Server.

3 Enter the address of an NTP server, if not already populated.




4 Click Synchronize Now, to get your system time set correctly.

5 If you want to make use of NTP permanently, enable Save NTP Configuration.

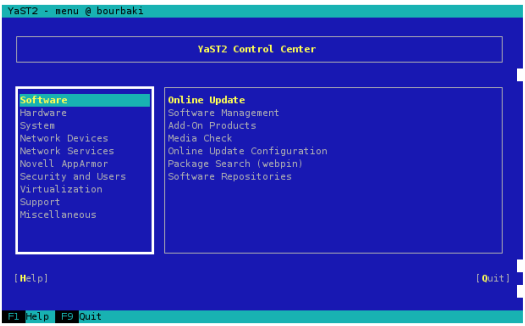
6 With the Configure button, you can open the advanced NTP configuration. For details, see Section “Configuring an NTP Client with YaST” (Chapter 13, Time Synchronization with NTP, ↑Reference).

7 Confirm your changes with Accept.

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 12.1 Main Window of YaST in Text Mode




When you start YaST in text mode, the YaST Control Center appears (see Figure 12.1). 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 highlighted. 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 [] + [highlighted_letter] to select a button directly instead of navigating there with [Tab]. Exit the YaST Control Center by pressing [] + [Q] or by selecting Quit and pressing [].

# 12.1 Navigation in Modules

The following description of the control elements in the YaST modules assumes that all function keys and [] key combinations work and are not assigned to different global functions. Read Section 12.2, “Restriction of Key Combinations”(page 149) for information about possible exceptions.

Navigation among Buttons and Selection Lists
: Use [Tab] to navigate among the buttons and frames containing selection lists. To navigate in reverse order, use [] + [Tab] or [] + [Tab] combinations.

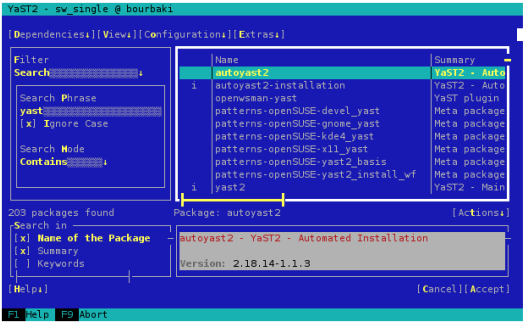
Navigation in Selection Lists
: Use the arrow keys ([↑] and [↓]) to navigate among the individual elements in an active frame containing a selection list. If individual entries within a frame exceed its width, use [] + [→] or [] + [←] to scroll horizontally to the right and left. Alternatively, use [] + [E] or [] + [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 []. Alternatively, radio buttons and check boxes can be selected directly with [] + [highlighted_letter]. In this case, you do not need to confirm with []. If you navigate to an item with [Tab], press [] 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 F key shortcuts 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, [] must be pressed after a selection in the navigation tree in order to show the selected dialog. This is an intentional behavior to save time consuming redraws when browsing through the navigation tree.

Figure 12.2 The Software Installation Module




# 12.2 Restriction of Key Combinations

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

Replacing [] with [Esc]
: [] shortcuts can be executed with [] instead of []. For example, [] – [H] replaces [] + [H]. (First press [], then press [H].)

Backward and Forward Navigation with [] + [F] and [] + [B]
: If the [] and [] combinations are occupied by the window manager or the terminal, use the combinations [] + [F] (forward) and [] + [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 [] key combinations and function keys should always be fully available on a pure text console.

# 12.3 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
```

## 12.3.1 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`.

### 12.3.2 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 new 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 9.1, “Using Zypper” (page 109).

### 12.3.3 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.
```

13 Setting Up Hardware Components with YaST

YaST allows you to configure hardware items at installation time as well as on an already-installed system. Configure audio hardware, printers or scanner support or learn which hardware components are connected to your computer by using the YaST Hardware Information module.

---

TIP: Graphics card, monitor, mouse and keyboard settings

Graphics card, monitor, mouse and keyboard can be configured with either KDE or GNOME tools.

---

# 13.1 Hardware Information

Use the YaST hardware information module if you want to know more about your hardware or if you need to find out details like vendor and model of a certain piece of hardware to be able to properly configure it.

1. Start YaST and click Hardware > Hardware Information. Hardware probing starts immediately and it will take some time until you see the hardware information tree in a separate window.

2. In the hardware information tree recursively click on the entries to expand the information about a specific device.

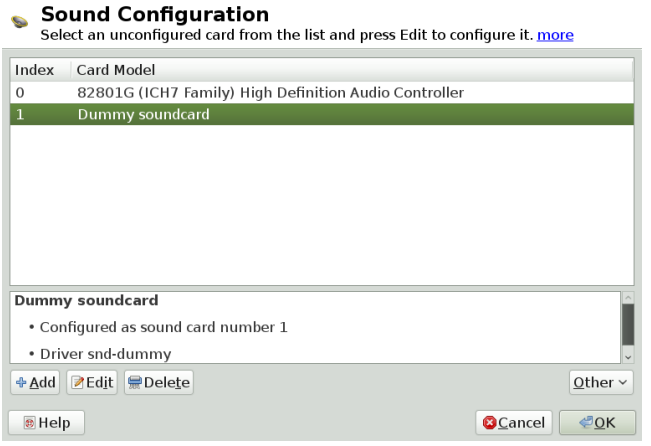
3. Click Save to File... to save the output to a file.

4. Click Close to leave the hardware information overview.

# 13.2 Setting Up Sound Cards

YaST detects most sound cards automatically and configures them with the appropriate values. If you want to change the default settings, or need to set up a sound card that could not be configured automatically, use the YaST sound module. There, you can also set up additional sound cards or switch their order.

To start the sound module, start YaST and click Hardware > Sound. Alternatively, start the Sound Configuration dialog directly by running `yast2 sound &` as user `root` from a command line.




The dialog shows all sound cards that could be detected.

Procedure 13.1 Configuring Sound Cards

If you have added a new sound card or YaST could not automatically configure an existing sound card, follow the steps below. For configuring a new sound card, you need to know your sound card vendor and model. If in doubt, refer to your sound card documentation for the required information. For a reference list of sound cards supported by ALSA with their corresponding sound modules, see http://www.alsa-project.org/main/index.php/Matrix:Main.

During configuration, you can choose between the following setup options:

Quick Automatic Setup
: You are not required to go through any of the further configuration steps—the sound card is configured automatically. You can set the volume or any options you want to change later.

Normal Setup
: Allows you to adjust the output volume and play a test sound during the configuration.

Advanced setup with possibility to change options
: For experts only. Allows you to customize all parameters of the sound card.

---

IMPORTANT: Advanced Configuration

Only use this option if you know exactly what your are doing. Otherwise leave the parameters untouched and use the normal or the automatic setup options.

---

1 Start the YaST sound module.

2 To configure a detected, but Not Configured sound card, select the respective entry from the list and click Edit.

   To configure a new sound card, click Add. Select your sound card vendor and model and click Next.

3 Choose one of the setup options and click Next.

4 If you have chosen Normal Setup, you can now Test your sound configuration and make adjustments to the volume. You should start at about ten percent volume to avoid damage to your hearing or the speakers.

5 If all options are set according to your wishes, click Next.

   The Sound Configuration dialog shows the newly configured or modified sound card.

6 To remove a sound card configuration that you no longer need, select the respective entry and click Delete.

7 Click OK to save the changes and leave the YaST sound module.

Procedure 13.2 Modifying Sound Card Configurations

1 To change the configuration of an individual sound card (for experts only!), select the sound card entry in the Sound Configuration dialog and click Edit.

   This takes you to the Sound Card Advanced Options where you can fine-tune a number of parameters. For more information, click Help.

2 To adjust the volume of an already configured sound card or to test the sound card, select the sound card entry in the Sound Configuration dialog and click Other. Select the respective menu item.

   ---

   NOTE: YaST Mixer

   The YaST mixer settings provide only basic options. They are intended for troubleshooting (for example, if the test sound is not audible). Access the YaST mixer settings from Other > Volume. For everyday use and fine-tuning of sound options, use the mixer applet provided by your desktop or the `alsasound` command line tool.

   ---

3 For playback of MIDI files, select Other > Start Sequencer.

4 When a supported sound card is detected (like a Creative `Soundblaster Live`, `Audigy` or `AWE` sound card), you can also install SoundFonts for playback of MIDI files:

   4a Insert the original driver CD-ROM into your CD or DVD drive.

4b Select Other > Install SoundFonts to copy SF2 SoundFonts™ to your hard disk. The SoundFonts are saved in the directory `/usr/share/sfbank/creative/`.

5 If you have configured more than one sound card in your system you can adjust the order of your sound cards. To set a sound card as primary device, select the sound card in the Sound Configuration and click Other > Set as the Primary Card. The sound device with index `0` is the default device and thus used by the system and the applications.

6 Per default, openSUSE uses the PulseAudio sound system. It is an abstraction layer that helps to mix multiple audio streams, bypassing any restrictions the hardware may have. To enable or disable the PulseAudio sound system, click Other > PulseAudio Configuration. If enabled, PulseAudio daemon is used to play sounds. Disable PulseAudio Support in case you want to use something else system-wide.

The volume and configuration of all sound cards are saved when you click OK and leave the YaST sound module. The mixer settings are saved to the file `/etc/asound.state`. The ALSA configuration data is appended to the end of the file `/etc/modprobe.d/sound` and written to `/etc/sysconfig/sound`.

## 13.3 Setting Up a Scanner

You can configure a USB or SCSI scanner with YaST. The `sane-backends` package contains hardware drivers and other essentials needed to use a scanner. Scanners connected to a parallel port cannot be configured with YaST. If you own a HP All-In-One device, see Section 13.3.1, “Configuring an HP All-In-One Device”(page 154), instructions on how to configure a network scanner are available at Section 13.3.3, “Scanning over the Network” (page 155).

Procedure 13.3 Configuring a USB or SCSI Scanner

1 Connect your USB or SCSI scanner to your computer and turn it on.

2 Start YaST and select Hardware > Scanner. YaST builds the scanner database and tries to detect your scanner model automatically.

If a USB or SCSI scanner is not properly detected, try Other > Restart Detection.

3 To activate the scanner select it from the list of detected scanners and click Edit.

4 Choose your model form the list and click Next and Finish.

5 Use Other > Test to make sure you have chosen the correct driver.

6 Leave the configuration screen with OK.

### 13.3.1 Configuring an HP All-In-One Device

An HP All-In-One device can be configured with YaST even if it is connected to the parallel port or is made available via the network. If you own a USB HP All-In-One device, start configuring as described in Procedure 13.3, “Configuring a USB or SCSI Scanner” (page 154). If it is detected properly and the Test succeeds, it is ready to use.

If your USB device is not properly detected, or your HP All-In-One device is connected to the parallel port or the network, run the HP Device Manager:

1. Start YaST and select Hardware > Scanner. YaST loads the scanner database.
2. Start the HP Device Manager with Other > Run hp-setup and follow the on-screen instructions. After having finished the HP Device Manager, the YaST scanner module automatically restarts the auto detection.
3. Test it by choosing Other > Test.
4. Leave the configuration screen with OK.

### 13.3.2 Sharing a Scanner over the Network

openSUSE allows the sharing of a scanner over the network. To do so, configure your scanner as follows:

1. Configure the scanner as described in Section 13.3, “Setting Up a Scanner” (page 154).
2. Choose Other > Scanning via Network.
3. Enter the hostnames of the clients (separated by a comma) that should be allowed to use the scanner under Server Settings > Permitted Clients for saned and leave the configuration dialog with OK.

### 13.3.3 Scanning over the Network

To use a scanner that is shared over the network, proceed as follows:

1. Start YaST and select Hardware > Scanner.
2. Open the network scanner configuration menu by Other > Scanning via Network.
3. Enter the hostname of the machine the scanner is connected to under Client Settings > Servers Used for the net Metadriver
4. Leave with OK. The network scanner is now listed in the Scanner Configuration window and is ready to use.

openSUSE® supports printing with many types of printers, including remote network printers. Printers can be configured manually or with YaST. For configuration instructions, refer to Section 14.4, “Setting Up a Printer” (page 159). 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 14.8, “Troubleshooting” (page 167).

CUPS (Common Unix Printing System) is the standard print system in openSUSE.

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 (like USB or parallel port) that is available on your hardware 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.

Standard Printers (Languages Like PCL and ESC/P)
: Although these printer languages are quite old, they are still undergoing expansion to address new features in printers. In the case of known printer languages, the print system can convert PostScript jobs to the respective printer language with the help of Ghostscript. This processing stage is referred to as 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 developing HPLIP (HP Linux Imaging and Printing), 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 14.8.1, “Printers without Standard Printer Language Support” (page 168) 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 dri-

vers available at production time. Accordingly, a printer currently rated as “perfectly supported” may not have had this status when the latest openSUSE 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-library/catalog.devices`
List of included drivers.

## 14.1 The Workflow of the Printing System

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 printer queue, and optionally, information for the filter, such as printer-specific options.

At least one dedicated printer 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.

## 14.2 Methods and Protocols for Connecting Printers

There are various possibilities for connecting a printer to the system. The configuration of the CUPS print system 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 in the Support Database at `http://en.opensuse.org/SDB:CUPS_in_a_Nutshell`.

---

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.

---

## 14.3 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. During the installation of openSUSE, many PPD files are preinstalled.

To configure a PostScript printer, the best approach is to get a suitable PPD file. Many PPD files are available in the package `manufacturer-PPDs`, which is automatically installed within the scope of the standard installation. See Section 14.7.2, “PPD Files in Various Packages” (page 166) and Section 14.8.2, “No Suitable PPD File Available for a PostScript Printer” (page 168).

New PPD files can be stored in the directory `/usr/share/cups/model/` or added to the print system with YaST as described in Section 14.4.1.1, “Adding Drivers with YaST” (page 161). 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 openSUSE 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.

## 14.4 Setting Up a Printer

YaST can be used to configure a local printer that is directly connected to your machine (normally with USB or parallel port) and to set up printing with network printers. It is also possible to share printers over the network. Further information about printing (general information, technical details, and troubleshooting) is available in Chapter 14, Printer Operation (page 157).

In YaST, click Hardware > Printer to start the printer module. By default it opens in the Printer Configurations view, displaying a list of all printers that are available and configured. This is especially useful when having access to a lot of printers via the network. From here you can also Print a Test Page and configure local printers.

### 14.4.1 Configuring Local Printers

Usually a local USB printer is automatically detected. There are two possible reasons why a USB printer is not automatically detected:

- The USB printer is switched off.
- The communication between printer and computer is not possible. Check the cable and the plugs to make sure that the printer is properly connected. If this is the case, the problem may not be printer-related, but rather a USB related problem.

Configuring a printer is basically a three-step process. First specify the connection type, then choose a driver, and name the printing queue for this setup.

For many printer models, several drivers are available. When configuring the printer, YaST defaults to the one marked `recommended` as a general rule. Normally it is not necessary to change the driver—the `recommended` one should produce the best results. However, if you want a color printer to print only in black and white, it is most convenient to use a driver that does not support color printing, for example. If you experience performance problems with a PostScript printer when printing graphics, it may help to switch from a PostScript driver to a PCL driver (provided your printer understands PCL).

If no driver for your printer is listed, you can try to select a generic driver with an appropriate standard language from the list. Refer to your printer's documentation to find out which language (the set of commands controlling the printer) your printer understands. If this does not work, refer to Section 14.4.1.1, “Adding Drivers with YaST”(page 161) for another possible solution.

A printer is never used directly, but always through a print queue. This ensures that simultaneous jobs can be queued and processed one after the other. Each printer queue is assigned to a specific driver, and a printer can have multiple queues. This makes it possible to set up a second queue on a color printer that prints black and white only, for example. Refer to Section 14.1, “The Workflow of the Printing System”(page 158) for more information about print queues.

Procedure 14.1 Adding a New Local Printer

1. Start the YaST printer module with Hardware > Printer
2. Click Add in the Printer Configurations screen
3. If your printer is already listed under `Specify the Connection`, proceed with the next step. Otherwise, try to Detect More or start the Connection Wizard.
4. Enter the vendor name and the model name into the input box under `Find and Assign a Driver` and click Search for.
5. Choose the driver marked as recommended that best matches your printer. If no suitable drivers is displayed
   - 5a check your search term
   - 5b broaden your search by clicking Find More
   - 5c add a driver as described in Section 14.4.1.1, “Adding Drivers with YaST” (page 161)
6. Specify the `Default paper size`
7. Enter a unique name for the printer queue in the Set Arbitrary Name field.
8. The printer is now configured with the default settings and ready to use. Click OK to return to the Printer Configurations view. The newly configured printer is now visible in the printers list.

#### 14.4.1.1 Adding Drivers with YaST

If no suitable driver is available in the Find and Assign a Driver dialog when adding a new printer, no PPD (PostScript Printer Description) file for your model is available. For more information about PPD files, refer to Section 14.3, “Installing the Software” (page 159).

Get PPD files directly from your printer vendor or from the driver CD of a PostScript printer. For details, see Section 14.8.2, “No Suitable PPD File Available for a PostScript Printer” (page 168). Conversely, you can also find PPD files at `http://www.linuxfoundation.org/en/OpenPrinting/`, the “OpenPrinting.org printer database”. When downloading PPD files from OpenPrinting.org, keep in mind that it always shows the latest Linux support status, which is not necessarily met by openSUSE.

Procedure 14.2 Adding a PPD file

1. Start the YaST printer module with Hardware > Printer
2. Click Add in the Printer Configurations screen
3. Click Driver Packages in the `Find and Assign a Driver` section
4. Enter the full path to the PPD file into the input box under `Make a Printer Description File Available` or choose the file from a dialog box by clicking on Browse
5. Click OK to return to the `Add New Printer Configuration` screen.
6. In order to directly use this PPD file, proceed as described in Procedure 14.1, “Adding a New Local Printer” (page 160). Otherwise, click Cancel.

#### 14.4.1.2 Editing a Local Printer Configuration

By editing an existing configuration for a local printer you can not only change basic settings as connection type and driver, but also adjust the default settings for paper size, resolution, media source, etc. You can change the identifier of the printer by altering the printer descriptions.

Procedure 14.3 Editing a Local Printer

1. Start the YaST printer module with Hardware > Printer
2. In the Printer Configurations screen, choose a local printer from the list and click Edit.
3. Change the connection type or the driver as described in Procedure 14.1, “Adding a New Local Printer” (page 160). This should only be necessary in case you have problems with the current configuration.
4. Make this printer the default by checking Default Printer.

5 Adjust the default settings by clicking All Options for the Current Driver. To change a setting, expand the list of options by clicking the relative + sign. Change the default by clicking on an option. Apply your changes by clicking OK

## 14.4.2 Configuring Printing via the Network with YaST

Network printers are not detected automatically. They must be configured manually using the YaST printer module. Depending on your network setup, you can print to a print server (CUPS, LPD, SMB, or IPX) or directly to a network printer (preferably via TCP). Access the configuration view for network printing by choosing Printing via Network from the left pane in the YaST printer module.

### 14.4.2.1 Using CUPS

In a Linux environment CUPS is usually used to print via the network. The simplest setup is to only print via a single CUPS server which can directly be accessed by all clients. Printing via more than one CUPS server requires a running local CUPS daemon that communicates with the remote CUPS servers.

Procedure 14.4 Printing via a Single CUPS server

1 Start the YaST printer module with Hardware > Printer

2 Launch the Print via Network screen from the left pane.

3 Check Do All Your Printing Directly via One Single CUPS Server and specify the name or IP address of the server.

4 Click Test Server to make sure you have chosen the correct name or IP address.

5 Click OK to return to the Printer Configurations screen. All printers available via the CUPS server are now listed.

Procedure 14.5 Printing via multiple CUPS servers

1 Start the YaST printer module with Hardware > Printer

2 Launch the Print via Network screen from the left pane.

3 Check Accept Printer Announcements from CUPS Servers

4 Specify which servers to use under `General Settings`. You may accept connections from all networks available, from the local network, or from specific hosts. If you choose the latter option, you need to specify the hostnames or IP addresses, as well.

5 Confirm by clicking OK and then Yes when asked to start a local CUPS server. After the server has started you will return to the Printer Configurations screen. Click Refresh list to see the printers detected by now. Click this button again, in case more printer are to be available.

#### 14.4.2.2 Using Print Servers other than CUPS

If your network offers print services via print servers other than CUPS, start the YaST printer module with Hardware > Printer and launch the Print via Network screen from the left pane. Start the Connection Wizard and choose the appropriate Connection type. Ask your network administrator for details on configuring a network printer in your environment.

### 14.4.3 Sharing Printers Over the Network

Printers managed by a local CUPS daemon can be shared over the network and so turn your machine into a CUPS server. Usually you share a printer by enabling CUPS' so-called “browsing mode”. If browsing is enabled, the local printer queues are made available on the network for listening to remote CUPS daemons. It is also possible to set up a dedicated CUPS server that manages all printing queues and can directly be accessed by remote clients. In this case it is not necessary to enable browsing.

Procedure 14.6 Sharing Printers

1. Start the YaST printer module with Hardware > Printer
2. Launch the Share Printers screen from the left pane.
3. Select Allow Remote Access. For more detailed configuration, additional options are available:
   - Check For computers within the local network and enable browsing mode by also checking Publish printers by default within the local network.
   - Add the network interface to be used by the CUPS server. If you want to share your printers via specified network interfaces, add those in the input box below.
   - In case you like to restrict access to your CUPS server to certain networks or IP addresses, specify these via the two input boxes.
4. Click OK to restart the CUPS server and return to the Printer Configurations screen.
5. Regarding CUPS and firewall settings, see http://en.opensuse.org/SDB:CUPS_and_SANE_Firewall_settings.

## 14.5 Network Printers

A network printer can support various protocols, some of them 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 have 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 of the 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 printer queue, is sent before the actual print data is sent. Therefore, a printer queue must be specified when configuring the LPD protocol. The implementations of diverse printer manufacturers are flexible enough to accept any name as the printer 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
```

## 14.5.1 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 parallel, and parameters. To determine valid device URIs on your system use the command `lpinfo -v | grep ":/"`:

```
# lpinfo -v | grep ":/"
direct usb://ACME/FunPrinter%20XL
direct parallel:/dev/lp0
```

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 parallel:/dev/lp0 -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`.

## 14.6 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`.

## 14.7 Special Features in openSUSE

A number of CUPS features have been adapted for openSUSE. Some of the most important changes are covered here.

### 14.7.1 CUPS and Firewall

After having performed a default installation of openSUSE, 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 Section “SuSEfirewall2” (Chapter 13, Masquerading and Firewalls, ↑Security Guide) and at http://en.opensuse.org/SDB:CUPS_and_SANE_Firewall_settings.

#### 14.7.1.1 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.

#### 14.7.1.2 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.

### 14.7.2 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 only have PostScript printers, normally you do not need the Foomatic PPD files in the `cups-drivers` package or the Gutenprint PPD files in the `gutenprint` package. Instead, the PPD files for your PostScript printers can be copied directly to `/usr/share/cups/model` (if they do not already exist in the `manufacturer-PPDs` package) to achieve an optimum configuration for your printers.

#### 14.7.2.1 CUPS PPD Files in the `cups` Package

The generic PPD files in the `cups` package have been complemented with adapted Foomatic PPD files for PostScript level 1 and level 2 printers:

- `/usr/share/cups/model/Postscript-level1.ppd.gz`
- `/usr/share/cups/model/Postscript-level2.ppd.gz`

#### 14.7.2.2 PPD Files in the `cups-drivers` Package

Normally, the Foomatic printer filter `foomatic-rip` is used together with Ghostscript for non-PostScript printers. Suitable Foomatic PPD files have the entries `*NickName: ... Foomatic/Ghostscript driver` and `*cupsFilter: ... foomatic-rip`. These PPD files are located in the `cups-drivers` package.

YaST generally prefers a `manufacturer-PPD` file. However, when no suitable `manufacturer-PPD` file exists, a Foomatic PPD file with the entry `*NickName: ... Foomatic ... (recommended)` is selected.

#### 14.7.2.3 Gutenprint PPD Files in the `gutenprint` Package

Instead of `foomatic-rip`, the CUPS filter `rastertogutenprint` from Gutenprint (formerly known as GIMP-Print) can be used for many non-PostScript printers. This filter and suitable Gutenprint PPD files are available in the `gutenprint` package. The Gutenprint PPD files are located in `/usr/share/cups/model/gutenprint/` and have the entries `*NickName: ... CUPS+Gutenprint` and `*cupsFilter: ... rastertogutenprint`.

#### 14.7.2.4 PPD Files from Printer Manufacturers in the `manufacturer-PPDs` Package

The `manufacturer-PPDs` package contains PPD files from printer manufacturers that are released under a sufficiently liberal license. PostScript printers should be configured with the suitable PPD file of the printer manufacturer, because this file enables the use of all functions of the PostScript printer. YaST prefers a PPD file from the `manufacturer-PPDs`. YaST cannot use a PPD file from the `manufacturer-PPDs` package if the model name does not match. This may happen if the `manufacturer-PPDs` package contains only one PPD file for similar models, like Funprinter 12xx series. In this case, select the respective PPD file manually in YaST.

## 14.8 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.

### 14.8.1 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 the fact that 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 due to new developments in the print system.

### 14.8.2 No Suitable PPD File Available for a PostScript Printer

If the `manufacturer-PPDs` package does 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.

### 14.8.3 Parallel Ports

The safest approach is to connect the printer directly to the first parallel port and to select the following parallel port settings in the BIOS:

- I/O address: `378` (hexadecimal)
- Interrupt: irrelevant
- Mode: `Normal`, `SPP`, or `Output Only`
- DMA: disabled

If the printer cannot be addressed on the parallel port despite these settings, enter the I/O address explicitly in accordance with the setting in the BIOS in the form `0x378` in `/etc/modprobe.conf`. If there are two parallel ports that are set to the I/O addresses `378` and `278` (hexadecimal), enter these in the form `0x378,0x278`.

If interrupt `7` is free, it can be activated with the entry shown in Example 14.1, "`/etc/modprobe.conf`: Interrupt Mode for the First Parallel Port" (page 169). Before activating the interrupt mode, check the file `/proc/interrupts` to see which interrupts are already in use. Only the interrupts currently being used are displayed. This may change depending on which hardware components are active. The interrupt for the parallel port must not be used by any other device. If you are not sure, use the polling mode with `irq=none`.

Example 14.1 /etc/modprobe.conf: Interrupt Mode for the First Parallel Port

```
alias parport_lowlevel parport_pc
options parport_pc io=0x378 irq=7
```

## 14.8.4 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 14.2, “Error Message from lpd” (page 170), the problem is caused by the remote lpd.

Example 14.2 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 14.3, “Broadcast from the CUPS Network Server” (page 170).

Example 14.3 Broadcast from the CUPS Network Server

```
ipp://192.168.2.202:631/printers/queue
```

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 have 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 14.5, “Network Printers” (page 163).

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 a number of 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.

## 14.8.5 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 is not able to print the printer-specific data, select a PPD file that is more suitable for the printer.

## 14.8.6 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 reenable printing with the command cupsenable.

## 14.8.7 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 as soon as 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`*

## 14.8.8 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 is unable to 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. For example, for a printer connected to the parallel port, the command `fuser -k /dev/lp0` can be used to terminate all processes that are still accessing the printer (more precisely: the parallel port).

4. Reset the printer completely by switching it off for some time. Then insert the paper and turn on the printer.

## 14.8.9 Debugging the CUPS Print System

Use the following generic procedure to locate problems in the CUPS print system:

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.

### 14.8.10 For More Information

Solutions to many specific problems are presented in the SUSE Support Database (http://en.opensuse.org/Portal:Support_database). Locate the relevant articles with a text search for `SDB:CUPS`.

The installation of additional fonts in openSUSE® is very easy. Simply copy the fonts to any directory located in the X11 font path . To the enable use of the fonts, the installation directory should be a subdirectory of the directories configured in `/etc/fonts/fonts.conf` or included into this file with `/etc/fonts/suse-font-dirs.conf`.

The following is an excerpt from `/etc/fonts/fonts.conf`. This file is the standard configuration file that should be appropriate for most configurations. It also defines the included directory `/etc/fonts/conf.d`. In this directory, all files or symbolic links starting with a two digit number are loaded by fontconfig. For a more detailed explanation of this functionality, have a look at `/etc/fonts/conf.d/README`.

```
<!-- Font directory list -->
<dir>/usr/share/fonts</dir>
<dir>/usr/X11R6/lib/X11/fonts</dir>
<dir>/opt/kde3/share/fonts</dir>
<dir>/usr/local/share/fonts</dir>
<dir>~/.fonts</dir>
```

`/etc/fonts/suse-font-dirs.conf` is automatically generated to pull in fonts that ship with (mostly third party) applications like OpenOffice.org, Java or Adobe Acrobat Reader. A typical entry would look like the following:

```
<dir>/usr/lib/Adobe/Reader9/Resource/Font</dir>
<dir>/usr/lib/Adobe/Reader9/Resource/Font/PFM</dir>
```

## 15.1 Adding Fonts

To install additional fonts system wide, manually copy the font files to a suitable directory (as `root`), such as `/usr/share/fonts/truetype`. Alternatively, the task can be performed with the KDE font installer in the System Settings. The result is the same.

Instead of copying the actual fonts, you can also create symbolic links. For example, you may want to do this if you have licensed fonts on a mounted Windows partition and want to use them. Subsequently, run `SuSEconfig --module fonts`.

`SuSEconfig --module fonts` executes the script `/usr/sbin/fonts-config`, which handles the font configuration. For more information on this script, refer to its manual page (`man fonts-config`).

The procedure is the same for bitmap fonts, TrueType and OpenType fonts, and Type 1 (PostScript) fonts. All these font types can be installed into any directory known to `fonts-config`.

You can upgrade an existing system without completely reinstalling it. There are two types of renewing the system or parts of it: updating individual software packages and upgrading the entire system. Updating individual packages is covered in Chapter 5, Installing or Removing Software (page 77) and Chapter 6, YaST Online Update (page 97). Two ways to upgrade the system are discussed in the following sections— see Section 16.1.3, “Upgrading with YaST”(page 178) and Section 16.1.4, “Distribution Upgrade with zypper” (page 179).

# 16.1 Upgrading the System

Software tends to “grow” from version to version. Therefore, take a look at the available partition space with `df` before updating. If you suspect you are running short of disk space, secure your data before you update and repartition your system. There is no general rule regarding how much space each partition should have. Space requirements depend on your particular partitioning profile, the software selected, and the version numbers of the system.

## 16.1.1 Preparations

Before upgrading, copy the old configuration files to a separate medium (such as removable hard disk or USB flash drive) to secure the data. This primarily applies to files stored in `/etc` as well as some of the directories and files in `/var`. You may also want to write the user data in `/home` (the `HOME` directories) to a backup medium. Back up this data as `root`. Only `root` has read permission for all local files.

Before starting your update, make note of the root partition. The command `df /` lists the device name of the root partition. In Example 16.1, “List with `df -h`”(page 177), the root partition to write down is `/dev/sda3` (mounted as `/`).

Example 16.1 List with df -h

```
Filesystem      Size  Used Avail Use% Mounted on
/dev/sda3        74G   22G   53G  29% /
udev            252M  124K  252M   1% /dev
/dev/sda5       116G  5.8G  111G   5% /home
/dev/sda1        39G  1.6G   37G   4% /windows/C
/dev/sda2       4.6G  2.6G  2.1G  57% /windows/D
```

## 16.1.2 Possible Problems

If you upgrade a default system from the previous version to this version, YaST works out the necessary changes and performs them. Depending on your customizations, some steps (or the entire upgrade procedure) may fail and you must resort to copying back your backup data. Check the following issues before starting the system update.

#### 16.1.2.1 Checking passwd and group in /etc

Before upgrading the system, make sure that `/etc/passwd` and `/etc/group` do not contain any syntax errors. For this purpose, start the verification utilities `pwck` and `grpck` as `root` to eliminate any reported errors.

#### 16.1.2.2 Shut Down Virtual Machines

If your machine serves as a VM Host Server for KVM or Xen, make sure to properly shut down all running VM Guests prior to the update. Otherwise you may not be able to acces the guests after the update.

#### 16.1.2.3 PostgreSQL

Before updating PostgreSQL (`postgres`), dump the databases. See the manual page of `pg_dump`. This is only necessary if you actually used PostgreSQL prior to your update.

### 16.1.3 Upgrading with YaST

Following the preparation procedure outlined in Section 16.1.1, “Preparations” (page 177), you can now upgrade your system:

1. Boot the system as for the installation, described in Section “System Start-Up for Installation” (Chapter 1, Installation with YaST, ↑Reference). In YaST, choose a language and select Update in the Installation Mode dialog. Do not select New Installation. Also add repositories to make sure to get all available software updated whenever possible. Find more information about installation repositories in Section “Add-On Products” (Chapter 1, Installation with YaST, ↑Reference).

2. YaST determines if there are multiple root partitions. If there is only one, continue with the next step. If there are several, select the right partition and confirm with Next (`/dev/sda3` was selected in the example in Section 16.1.1, “Preparations” (page 177)). YaST reads the old `fstab` on this partition to analyze and mount the file systems listed there.

   ---

   WARNING: Persistent Device Names

   All entries in `/etc/fstab` that specify partitions to be mounted using the kernel-device name must be changed to any of the other supported methods prior to performing an update. Kernel-device names are not persistent and are therefore unreliable for use during the update process. This can be done using the YaST System Partitioner by changing the method used in the fstab options settings.

   ---

3. Check the previously used repositories, if there are any. Enable all the repositories you still want to use and from where you want to update third-party software. Click the Toggle Status for every list item, if appropriate.

4. If you added repositories during the upgrade procedure as recommended above, you now can activate those you are actually interested in.

5 In the Installation Settings dialog, adjust the settings according to your requirements. Normally, you can leave the default settings untouched. If you intend to enhance your system, however, check the packages and patterns offered in the Packages and Update Options submenus, or add support for additional languages.

   You also have the possibility to make backups of various system components. Selecting backups slows down the upgrade process. Use this option if you do not have a recent system backup.

6 Confirm the upgrade by clicking Start Update.

Once the basic upgrade installation is finished, YaST reboots the system. Finally, YaST updates the remaining software, if any and displays the release notes, if wanted.

## 16.1.4 Distribution Upgrade with zypper

With the `zypper` command line utility you can upgrade to the next version of the distribution. Most importantly, you can initiate the system upgrade process from within the running system.

This feature is attractive for advanced users who want to run remote upgrades or upgrades on many similarly configured systems.

### 16.1.4.1 Before Starting the Upgrade with zypper

To avoid unexpected errors during the upgrade process using `zypper`, minimize risky constellations.

Upgrade from the previous version (e.g., 11.3) to this version (11.4)—do not skip any minor version inbetween (this means, do not upgrade from 11.2 or earlier to 11.4 in one go). Make sure all available 11.3 online updates are successfully applied.

Close as many applications and unneeded services as possible and log out all regular users.

Disable third party or openSUSE Build Server repositories before starting the upgrade, or lower the priority of these repositories to make sure packages from the default system repositories will get preference. Enable them again after the upgrade and edit their version string to match the version number of the distribution of the upgraded now running system.

For more information, see `http://en.opensuse.org/SDB:System_upgrade`.

### 16.1.4.2 The Upgrade Procedure

---

WARNING: Check Your System Backup

Before actually starting the upgrade procedure, check that your system backup is up-to-date and restorable. This is especially important because you must enter many of the following steps manually.

---

1 Run the online update to make sure the software management stack is up-to-date. For more information, see Chapter 6, YaST Online Update (page 97).

2 Configure the repositories you want to use as an update source. Getting this right is essential. Either use YaST (see Section 5.4, “Managing Software Repositories and Services” (page 88)) or `zypper` (see Section 9.1, “Using Zypper” (page 109)). The name of the repositories as used in the following steps could vary a little bit depending on your customizations.

To view your current repositories enter:

```
zypper lr -u
```

---

TIP: zypper command names

zypper supports long and short command names. For example, you can abbreviate `zypper install` as `zypper in`. In the following text, the short variant is used.

---

2a Increase the version number of the system repositories from 11.3 to 11.4; add the new 11.4 repositories with commands such as:

```
server=http://download.opensuse.org
new_ver=11.4
zypper ar $server/distribution/$new_ver/repo/oss/
 openSUSE-$new_ver-Oss
zypper ar $server/update/$new_ver/ openSUSE-
$new_ver-Update
```

And remove the old repositories:

```
old_ver=11.3
zypper rr openSUSE-$old_ver-Oss
zypper rr openSUSE-$old_ver-Update
```

2b Disable third party repositories or other openSUSE Build Server repositories, because `zypper dup` is guaranteed to work with the default repositories only (replace *`repo-alias`* with the name of the repository you want to disable):

```
zypper mr -d repo-alias
```

Alternatively, you can lower the priority of these repositories.

---

NOTE: Handling of Unresolved Dependencies

`zypper dup` will remove all packages having unresolved dependencies, but it keeps packages of disabled repositories as long as their dependencies are satisfied.

---

`zypper dup` ensures that all installed packages come from one of the available repositories. It does not consider the version, architecture, or vendor of the installed packages; thus it emulates a fresh installation. Packages that are no longer available in the repositories are considered orphaned. Such packages get uninstalled if their dependencies can not be satisfied. If they can be satisfied, such packages stay installed.

2c Once done, check your repository configuration with:

```
zypper lr -d
```

3 Refresh local metadata and repository contents with `zypper ref`.

4 Pull in zypper and the package management stack from the 11.4 repository with `zypper up zypper`.

5 Run the actual distribution upgrade with `zypper dup`. You are asked to confirm the license.

6 Perform basic system configuration with `SuSEconfig`.

7 Reboot the system with `shutdown -r now`.

### 16.1.5 Updating Individual Packages

Regardless of your overall updated environment, you can always update individual packages. From this point on, however, it is your responsibility to ensure that your system remains consistent.

Use the YaST software management tool to update packages as described in Chapter 5, Installing or Removing Software (page 77). Select components from the YaST package selection list according to your needs. If a newer version of a package exists, the version numbers of the installed and the available versions are listed in blue color in the Installed (Available) column. If you select a package essential for the overall operation of the system, YaST issues a warning. Such packages should be updated only in the update mode. For example, many packages contain shared libraries. Updating these programs and applications in the running system may lead to system instability.

## 16.2 For More Information

Problems and special issues of the various versions are published online as they are identified. See the links listed below. Important updates of individual packages can be accessed using the YaST Online Update. For more information, see Chapter 6, YaST Online Update (page 97).

Refer to the `Product highlights` (`http://en.opensuse.org/Product_highlights`) and the `Bugs` article in the openSUSE wiki at `http://`

en.opensuse.org/openSUSE:Most_annoying_bugs for information about recent changes and issues.

# Part IV. The Bash Shell

When working with Linux these days, you can communicate with the system almost without ever requiring a command line interpreter (the shell). After booting your Linux system, you are usually directed to a graphical user interface that guides you through the login process and the following interactions with the operating system. The graphical user interface in Linux (the X Window System or X11) is initially configured during installation. Both KDE and GNOME desktop (and other window managers you can install) use it for interaction with the user.

Nevertheless, it is useful to have some basic knowledge of working with a shell because you might encounter situations where the graphical user interface is not available. For example, if some problem with the X Window System occurs. If you are not familiar with a shell, you might feel a bit uncomfortable at first when entering commands, but the more you get used to it, the more you will realize that the command line is often the quickest and easiest way to perform some daily tasks.

For UNIX or Linux several shells are available which differ slightly in behavior and in the commands they accept. The default shell in openSUSE® is Bash (GNU Bourne-Again Shell).

The following sections will guide you through your first steps with the Bash shell and will show you how to complete some basic tasks via the command line. If you are interested in learning more or rather feel like a shell “power user” already, refer to Chapter 18, Bash and Bash Scripts (page 217).

## 17.1 Starting a Shell

Basically, there are two different ways to start a shell from the graphical user interface which usually shows after you have booted your computer:

- you can leave the graphical user interface or
- you can start a terminal window within the graphical user interface.

While the first option is always available, you can only make use of the second option when you are already logged in to a desktop such as KDE or GNOME. Whichever way you choose, there is always a way back and you can switch back and forth between the shell and the graphical user interface.

If you want to give it a try, press [] + [] + [F2] to leave the graphical user interface. The graphical user interface disappears and you are taken to a shell which prompts you to log in. Type your username and press []. Then type your password and press []. The prompt now changes and shows some useful information as in the following example:

❶ ❷ ❸
```
tux@linux:~>
```

❶ Your login.
❷ The hostname of your computer.
❸ Path to the current directory. Directly after login, the current directory usually is your home directory, indicated by the ~ symbol (tilde) .

When you are logged in at a remote computer the information provided by the prompt always shows you which system you are currently working on.

When the cursor is located behind this prompt, you can pass commands directly to your computer system. For example, you can now enter `ls -l` to list the contents of the current directory in a detailed format. If this is enough for your first encounter with the shell and you want to go back to the graphical user interface, you should log out from your shell session first. To do so, type `exit` and press []. Then press [] + [F7] to switch back to the graphical user interface. You will find your desktop and the applications running on it unchanged.

When you are already logged in to the GNOME or the KDE desktop and want to start a terminal window within the desktop, press [] + [F2] and enter `konsole` (for KDE) or `gnome-terminal` (for GNOME). This opens a terminal window on your desktop. As you are already logged in to your desktop, the prompt shows information about your system as described above. You can now enter commands and execute tasks just like in any shell which runs parallel to your desktop. To switch to another application on the desktop just click on the corresponding application window or select it from the taskbar of your panel. To close the terminal window press [] + [F4].

## 17.2 Entering Commands

As soon as the prompt appears on the shell it is ready to receive and execute commands. A command can consist of several elements. The first element is the actual command, followed by parameters or options. You can type a command and edit it by using the following keys: [], [], [], [], [] (Backspace), [], and []. You can correct typing errors or add options. The command is not executed until you press [].

---

IMPORTANT: No News Is Good News

The shell is not verbose: in contrast to some graphical user interfaces, it usually does not provide confirmation messages when commands have been executed. Messages only appear in case of problems or errors —or if you explicitly ask for them by executing a command with a certain option.

Also keep this in mind for commands to delete objects. Before entering a command like `rm` (without any option) for removing a file, you should know if you really want to get rid of the object: it will be deleted irretrievably, without confirmation.

---

### 17.2.1 Using Commands without Options

In Section 17.5.1, “Permissions for User, Group and Others” (page 194) you already got to know one of the most basic commands: `ls`, which used to list the contents of

a directory. This command can be used with or without options. Entering the plain `ls` command shows the contents of the current directory:

```
tux@knox:~> ls
bin Desktop Documents public_html tux.txt
tux@knox:~>
```

Files in Linux may have a file extension or a suffix, such as `.txt`, but do not need to have one. This makes it difficult to differentiate between files and folders in this output of the `ls`. By default, the colors in the Bash shell give you a hint: directories are usually shown in blue, files in black.

## 17.2.2 Using Commands with Options

A better way to get more details about the contents of a directory is using the `ls` command with a string of options. Options modify the way a command works so that you can get it to carry out specific tasks. Options are separated from the command with a blank and are usually prefixed with a hyphen. The `ls -l` command shows the contents of the same directory in full detail (long listing format):

```
tux@knox:~> ls -l
drwxr-xr-x 1 tux users     48 2006-06-23 16:08 bin
drwx---r-- 1 tux users  53279 2006-06-21 13:16 Desktop
drwx------ 1 tux users    280 2006-06-23 16:08 Documents
drwxr-xr-x 1 tux users  70733 2006-06-21 09:35 public_html
-rw-r--r-- 1 tux users  47896 2006-06-21 09:46 tux.txt
tux@knox:~>
```

This output shows the following information about each object:

```
drwxr-xr-x❶ 1❷ tux❸ users❹ 48❺ 2006-06-23 16:08❻ bin❼
```

❶ Type of object and access permissions. For further information, refer to Section 17.5.1, “Permissions for User, Group and Others” (page 194).

❷ Number of hard links to this file.

❸ Owner of the file or directory. For further information, refer to Section 17.5.1, “Permissions for User, Group and Others” (page 194).

❹ Group assigned to the file or directory. For further information, refer to Section 17.5.1, “Permissions for User, Group and Others” (page 194).

❺ File size in bytes.

❻ Date and time of the last change.

❼ Name of the object.

Usually, you can combine several options by prefixing only the first option with a hyphen and then write the others consecutively without a blank. For example, if you want to see all files in a directory in long listing format, you can combine the two options `-l` and `-a` (show all files) for the `ls` command. Executing `ls -la` shows also hidden files in the directory, indicated by a dot in front (for example, `.hiddenfile`).

The list of contents you get with `ls` is sorted alphabetically by filenames. But like in a graphical file manager, you can also sort the output of `ls -l` according to various criteria such as date, file extension or file size:

- For date and time, use `ls -lt` (displays newest first).
- For extensions, use `ls -lx` (displays files with no extension first).
- For file size, use `ls -lS` (displays largest first).

To revert the order of sorting, add `-r` as an option to your `ls` command. For example, `ls -lr` gives you the contents list sorted in reverse alphabetical order, `ls -ltr` shows the oldest files first. There are lots of other useful options for `ls`. In the following section you will learn how to investigate them.

### 17.2.3 Getting Help

Nobody is expected to know all options of all commands by heart. If you remember the command name but are not sure about the options or the syntax of the command, choose one of the following possibilities:

`--help` option
: If you only want to look up the options of a certain command, try entering the command followed by a space and `--help`. This `--help` option exists for many commands. For example, `ls --help` displays all the options for the `ls` command.

Manual Pages
: To learn more about the various commands, you can also use the manual pages. Manual pages also give a short description of what the command does. They can be accessed with `man` followed by the name of the command, for example, `man ls`.

  The man pages are displayed directly in the shell. To navigate them, move up and down with [] and []. Move between the beginning and the end of a document with [] and []. End this viewing mode by pressing [Q]. Learn more about the `man` command itself with `man man`.

Info Pages
: Info pages usually provide even more information about commands. 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 [] to move forward and [] to move backwards. Within a node, you can also browse with [] and [] but only [] and [] will take you also to the previous or subsequent node. Like for the man pages, press [Q] to end the viewing mode.

Note that man pages and info pages do not exist for all commands. Sometimes both are available (usually for key commands), sometimes only a man page or an info page exists, and sometimes neither of them are available.

### 17.2.4 Bash Shortcut Keys

After having entered several commands, your shell will begin to fill up with all sorts of commands and the corresponding outputs. In the following table, find some useful shortcut keys for navigating and editing in the shell.

| Shortcut Key | Function |
|---|---|
| [] + [L] | Clears the screen and moves the current line to the top of the page. |
| [] + [C] | Aborts the command which is currently being executed. |
| [] + [] | Scrolls upwards. |
| [] + [] | Scrolls downwards. |
| [] + [U] | Deletes from cursor position to start of line. |
| [] + [K] | Deletes from cursor position to the end of line. |
| [] + [D] | Closes the shell session. |
| [], [] | Browses in the history of executed commands. |

## 17.3 Working with Files and Directories

To address a certain file or directory, you must specify the path leading to that directory or file. As you may know from MS DOS or Mac OS already, there are two ways to specify a path:

Absolute Path
: Enter the entire path from the root directory to the relevant file or directory.

Relative Path
: Enter a path to the relevant file or directory by using the current directory as a starting point. This implies to give the levels you have to move up or down in the file system tree to reach the target directory of file, starting from the current directory.

Paths contain filenames, directories or both, separated by slashes. Absolute paths always start with a slash. Relative paths do not have a slash at the beginning, but can have one or two dots.

When entering commands, you can choose either way to specify a path, depending on your preferences or the amount of typing, both will lead to the same result. To change directories, use the `cd` command and specify the path to the directory.

NOTE: Handling Blanks in Filenames or Directory Names

If a filename or the name of a directory contains a space, either escape the space using a back slash (`\`) in front of the blank or enclose the filename in single quotes. Otherwise Bash interprets a filename like `My Documents` as the names of two files or directories, `My` and `Documents` in this case.

When specifying paths, the following “shortcuts” can save you a lot of typing:

- The tilde symbol (`~`) is a shortcut for home directories. For example, to list the contents of your home directory, use `ls ~`. To list the contents of another user's home directory, enter `ls ~username` (or course, this will only work if you have permission to view the contents, see Section 17.5, “File Access Permissions” (page 194)). For example, entering `ls ~tux` would list the contents of the home directory of a user named `tux`. You can use the tilde symbol as shortcut for home directories also if you are working in a network environment where your home directory may not be called `/home` but can be mapped to any directory in the file system.

  From anywhere in the file system, you can reach your home directory by entering `cd ~` or by simply entering `cd` without any options.

- When using relative paths, refer to the current directory with a dot (`.`). This is mainly useful for commands such as `cp` or `mv` by which you can copy or move files and directories.

- The next higher level in the tree is represented by two dots (`..`). In order to switch to the parent directory of your current directory, enter `cd ..`, to go up two levels from the current directory enter `cd ../..` etc.

To apply your knowledge, find some examples below. They address basic tasks you may want to execute with files or folders using Bash.

## 17.3.1 Examples for Working with Files and Directories

Suppose you want to copy a file located somewhere in your home directory to a subdirectory of `/tmp` that you need to create first.

Procedure 17.1 Creating and Changing Directories

From your home directory create a subdirectory in `/tmp`:

1 Enter

```
mkdir /tmp/test
```

`mkdir` stands for “make directory”. This command creates a new directory named `test` in the `/tmp` directory. In this case, you are using an absolute path to create the `test` directory.

2 To check what happened, now enter

```
ls -l /tmp
```

The new directory `test` should appear in the list of contents of the `/tmp` directory.

3 Switch to the newly created directory with

```
cd /tmp/test
```

Procedure 17.2 Creating and Copying Files

Now create a new file in a subdirectory of your home directory and copy it to `/tmp/test`. Use a relative path for this task.

---

IMPORTANT: Overwriting of Existing Files

Before copying, moving or renaming a file, check if your target directory already contains a file with the same name. If yes, consider changing one of the filenames or use `cp` or `mv` with options like `-i`, which will prompt before overwriting an existing file. Otherwise Bash will overwrite the existing file without confirmation.

---

1 To list the contents of your home directory, enter

```
ls -l ~
```

It should contain a subdirectory called `Documents` by default. If not, create this subdirectory with the `mkdir` command you already know:

```
mkdir ~/Documents
```

2 To create a new, empty file named `myfile.txt` in the `Documents` directory, enter

```
touch ~/Documents/myfile.txt
```

Usually, the `touch` command updates the modification and access date for an existing file. If you use `touch` with a filename which does not exist in your target directory, it creates a new file.

3 Enter

```
ls -l ~/Documents
```

The new file should appear in the list of contents.

4 To copy the newly created file, enter

```
cp ~/Documents/myfile.txt .
```

Do not forget the dot at the end.

This command tells Bash to go to your home directory and to copy `myfile.txt` from the `Documents` subdirectory to the current directory, `/tmp/test`, without changing the name of the file.

5 Check the result by entering

```
ls -l
```

The file `myfile.txt` should appear in the list of contents for `/tmp/test`.

Procedure 17.3 Renaming and Removing Files or Directories

Now suppose you want to rename `myfile.txt` into `tuxfile.txt`. Finally you decide to remove the renamed file and the `test` subdirectory.

1 To rename the file, enter

```
mv myfile.txt tuxfile.txt
```

2 To check what happened, enter

```
ls -l
```

Instead of `myfile.txt`, `tuxfile.txt` should appear in the list of contents.

`mv` stands for `move` and is used with two options: the first option specifies the source, the second option specifies the target of the operation. You can use `mv` either

- to rename a file or a directory,
- to move a file or directory to a new location or
- to do both in one step.

3 Coming to the conclusion that you do not need the file any longer, you can delete it by entering

```
rm tuxfile.txt
```

Bash deletes the file without any confirmation.

4 Move up one level with `cd ..` and check with

```
ls -l test
```

if the `test` directory is empty now.

5 If yes, you can remove the `test` directory by entering

```
rmdir test
```

## 17.4 Becoming Root

root, also called the superuser, has privileges which authorize him to access all parts of the system and to execute administrative tasks. He or she has the unrestricted capacity to make changes to the system and has unlimited access to all files. Therefore performing some administrative tasks or running certain programs such as YaST requires root permissions.

## 17.4.1 Using su

In order to temporarily become root in a shell, proceed as follows:

1. Enter su. You are prompted for the root password.
2. Enter the password. If you mistyped the root password, the shell displays a message. In this case, you have to re-enter su before retyping the password. If your password is correct, a hash symbol # appears at the end of the prompt, signaling that you are acting as root now.
3. Execute your task. For example, transfer ownership of a file to a new user which only root is allowed to do:
   ```
   chown wilber kde_quick.xml
   ```
4. After having completed your tasks as root, switch back to your normal user account. To do so, enter
   ```
   exit
   ```
   The hash symbol disappears and you are acting as “normal” user again.

## 17.4.2 Using sudo

Alternatively, you can also use sudo (superuser “do”) to execute some tasks which normally are for roots only. With sudo, administrators can grant certain users root privileges for some commands. Depending on the system configuration, users can then run root commands by entering their normal password only. Due to a timestamp function, users are only granted a “ticket” for a restricted period of time after having entered their password. The ticket usually expires after a few minutes. In openSUSE, sudo requires the root password by default (if not configured otherwise by your system administrator).

For users, sudo is convenient as it prevents you from switching accounts twice (to root and back again). To change the ownership of a file using sudo, only one command is necessary instead of three:

```
sudo chown wilber kde_quick.xml
```

After you have entered the password which you are prompted for, the command is executed. If you enter a second root command shortly after that, you are not prompted for the password again, because your ticket is still valid. After a certain amount of time, the ticket automatically expires and the password is required again. This also prevents unauthorized persons from gaining root privileges in case a user forgets to switch back to his normal user account again and leaves a root shell open.

# 17.5 File Access Permissions

In Linux, objects such as files or folders or processes generally belong to the user who created or initiated them. There are some exceptions to this rule. For more information about the exceptions, refer to Chapter 9, Access Control Lists in Linux (↑Security Guide). The group which is associated with a file or a folder depends on the primary group the user belongs to when creating the object.

When you create a new file or directory, initial access permissions for this object are set according to a predefined scheme. As an owner of a file or directory, you can change the access permissions for this object. For example, you can protect files holding sensitive data against read access by other users and you can authorize the members of your group or other users to write, read, or execute several of your files where appropriate. As `root`, you can also change the ownership of files or folders.

## 17.5.1 Permissions for User, Group and Others

Three permission sets are defined for each file object on a Linux system. These sets include the read, write, and execute permissions for each of three types of users—the owner, the group, and other users.

The following example shows the output of an `ls -l` command in a shell. This command lists the contents of a directory and shows the details for each file and folder in that directory.

Example 17.1 Access Permissions For Files and Folders

```
-rw-r----- 1 tux users      0 2006-06-23 16:08 checklist.txt
-rw-r--r-- 1 tux users  53279 2006-06-21 13:16 gnome_quick.xml
-rw-rw---- 1 tux users      0 2006-06-23 16:08 index.htm
-rw-r--r-- 1 tux users  70733 2006-06-21 09:35 kde-start.xml
-rw-r--r-- 1 tux users  47896 2006-06-21 09:46 kde_quick.xml
drwxr-xr-x 2 tux users     48 2006-06-23 16:09 local
-rwxr--r-- 1 tux users 624398 2006-06-23 15:43 tux.sh
```

As shown in the third column, all objects belong to user `tux`. They are assigned to the group `users` which is the primary group the user `tux` belongs to. To retrieve the access permissions the first column of the list must be examined more closely. Let's have a look at the file `kde-start.xml`:

| Type | User Permissions | Group Permissions | Permissions for Others |
|---|---|---|---|
| - | rw- | r-- | r-- |

The first column of the list consists of one leading character followed by nine characters grouped in three blocks. The leading character indicates the file type of the object: in this case, the hyphen (`-`) shows that `kde-start.xml` is a file. If you find the character `d` instead, this shows that the object is a directory, like `local` in Example 17.1, “Access Permissions For Files and Folders” (page 194).

The next three blocks show the access permissions for the owner, the group and other users (from left to right). Each block follows the same pattern: the first position shows read permissions (`r`), the next position shows write permissions (`w`), the last one shows execute permission (`x`). A lack of either permission is indicated by `-`. In our example, the owner of `kde-start.xml` has read and write access to the file but cannot execute it. The `users` group can read the file but cannot write or execute it. The same holds true for the other users as shown in the third block of characters.

## 17.5.2 Files and Folders

Access permissions have a slightly different impact depending on the type of object they apply to: file or directory. The following table shows the details:

Table 17.1 Access Permissions For Files And Directories

| Access Permission | File | Folder |
|---|---|---|
| Read (r) | Users can open and read the file. | Users can view the contents of the directory. Without this permission, users cannot list the contents of this directory with `ls -l`, for example. However, if they only have execute permission for the directory, they can nevertheless access certain files in this directory if they know of their existence. |
| Write (w) | Users can change the file: They can add or drop data and can even delete the contents of the file. However, this does not include the permission to remove the file completely from the directory as long as they do not have write permissions for the directory where the file is located. | Users can create, rename or delete files in the directory. |
| Execute (x) | Users can execute the file. This permission is only relevant for files like programs or shell | Users can change into the directory and execute files there. If they do not have read access to |

| Access Permission | File | Folder |
|---|---|---|
| | scripts, not for text files. If the operating system can execute the file directly, users do not need read permission to execute the file. However, if the file must me interpreted like a shell script or a perl program, additional read permission is needed. | that directory they cannot list the files but can access them nevertheless if they know of their existence. |

Note that access to a certain file is always dependent on the correct combination of access permissions for the file itself and the directory it is located in.

### 17.5.3 Modifying File Permissions

In Linux, objects such as files or folder or processes generally belong to the user who created or initiated them. The group which is associated with a file or a folder depends on the primary group the user belongs to when creating the object. When you create a new file or directory, initial access permissions for this object are set according to a predefined scheme. For further details refer to Section 17.5, “File Access Permissions” (page 194).

As the owner of a file or directory (and, of course, as `root`), you can change the access permissions to this object.

To change object attributes like access permissions of a file or folder, use the `chmod` command followed by the following parameters:

- the users for which to change the permissions,
- the type of access permission you want to remove, set or add and
- the files or folders for which you want to change permissions separated by spaces.

The users for which you can change file access permissions fall into the following categories: the owner of the file (user, `u`), the group that own the file (group, `g`) and the other users (others, `o`). You can add, remove or set one or more of the following permissions: read, write or execute.

As `root`, you can also change the ownership of a file: with the command `chown` (change owner) you can transfer ownership to a new user.

#### 17.5.3.1 Examples for Changing Access Permissions and Ownership

The following example shows the output of an `ls -l` command in a shell.

Example 17.2 Access Permissions For Files and Folders

```
-rw-r----- 1 tux users      0 2006-06-23 16:08 checklist.txt
-rw-r--r-- 1 tux users  53279 2006-06-21 13:16 gnome_quick.xml
-rw-rw---- 1 tux users      0 2006-06-23 16:08 index.htm
-rw-r--r-- 1 tux users  70733 2006-06-21 09:35 kde-start.xml
-rw-r--r-- 1 tux users  47896 2006-06-21 09:46 kde_quick.xml
drwxr-xr-x 2 tux users     48 2006-06-23 16:09 local
-r-xr-xr-x 1 tux users 624398 2006-06-23 15:43 tux.jpg
```

In the example above, user `tux` owns the file `kde-start.xml` and has read and write access to the file but cannot execute it. The `users` group can read the file but cannot write or execute it. The same holds true for the other users as shown by the third block of characters.

Procedure 17.4 Changing Access Permissions

Suppose you are `tux` and want to modify the access permissions to your files:

1. If you want to grant the `users` group also write access to `kde-start.xml`, enter

   ```
   chmod g+w kde-start.xml
   ```

2. To grant the `users` group and other users write access to `kde-start.xml`, enter

   ```
   chmod go+w kde-start.xml
   ```

3. To remove write access for all users, enter

   ```
   chmod -w kde-start.xml
   ```

   If you do not specify any kind of users, the changes apply to all users— the owner of the file, the owning group and the others. Now even the owner `tux` does not have write access to the file without first reestablishing write permissions.

4. To prohibit the `users`group and others to change into the directory `local`, enter

   ```
   chmod go-x local
   ```

5. To grant others write permissions for two files, for `kde_quick.xml` and `gnome_quick.xml`, enter

   ```
   chmod o+w  kde_quick.xml gnome_quick.xml
   ```

Procedure 17.5 Changing Ownership

Suppose you are `tux` and want to transfer the ownership of the file `kde_quick.xml` to an other user, named `wilber`. In this case, proceed as follows:

1. Enter the username and password for `root`.

2 Enter

```
chown wilber kde_quick.xml
```

3 Check what happened with

```
ls -l kde_quick.xml
```

You should get the following output:

```
-rw-r--r-- 1 wilber users  47896 2006-06-21 09:46
 kde_quick.xml
```

4 If the ownership is set according to your wishes, switch back to your normal user account.

## 17.6 Useful Features of the Shell

As you probably noticed in the examples above, entering commands in Bash can include a lot of typing. In the following, get to know some features of the Bash that can make your work a lot easier and save a lot of typing.

History

By default, Bash “remembers” commands you have entered. This feature is called history. You can browse through commands that have been entered before, select one you want to repeat and then execute it again. To do so, press [] repeatedly until the desired command appears at the prompt. To move forward through the list of previously entered commands, press []. For easier repetition of a certain command from Bash history, just type the first letter of the command you want to repeat and press [].

You can now edit the selected command (for example, change the name of a file or a path), before you execute the command by pressing []. To edit the command line, just move the cursor to the desired position using the arrow keys and start typing.

You can also search for a certain command in the history. Press [] + [R] to start an incremental search function. showing the following prompt:

```
(reverse-i-search)`':
```

Just type one or several letters from the command you are searching for. Each character you enter narrows down the search. The corresponding search result is shown on the right side of the colon whereas your input appears on the left of the colon. To accept a search result, press []. The prompt now changes to its normal appearance and shows the command you chose. You can now edit the command or directly execute it by pressing [].

Completion

Completing a filename or directory name to its full length after typing its first letters is another helpful feature of Bash. To do so, type the first letters then press [] (Tabulator). If the filename or path can be uniquely identified, it is completed at once and the cursor moves to the end of the filename. You can then enter the next option

of the command, if necessary. If the filename or path cannot be uniquely identified (because there are several filenames starting with the same letters), the filename or path is only completed up to the point where it becomes ambiguous again. You can then obtain a list of them by pressing [] a second time. After this, you can enter the next letters of the file or path then try completion again by pressing []. When completing filenames and paths with the help of [], you can simultaneously check whether the file or path you want to enter really exists (and you can be sure of getting the spelling right).

Wild Cards

You can replace one or more characters in a filename with a wild card for pathname expansion. Wild cards are characters that can stand for other characters. There are three different types of these in Bash:

| Wild Card | Function |
|---|---|
| `?` | Matches exactly one arbitrary character |
| `*` | Matches any number of characters |
| `[set]` | Matches one of the characters from the group specified inside the square brackets, which is represented here by the string *`set`*. |

## 17.6.1 Examples For Using History, Completion and Wildcards

The following examples illustrate how to make use of these convenient features of Bash.

Procedure 17.6 Using History and Completion

If you already did the example Section 17.3.1, “Examples for Working with Files and Directories” (page 190) your shell buffer should be filled with commands which you can retrieve using the history function.

1. Press [] repeatedly until `cd ~` appears.
2. Press [] to execute the command and to switch to your home directory.

   By default, your home directory contains two subdirectories starting with the same letter, `Documents` and `Desktop`.
3. Enter `cd D` and press [].

   Nothing happens since Bash cannot identify to which one of the subdirectories you want to change.

4 Press [] again to see the list of possible choices:

```
tux@knox:~> cd D Desktop/   Documents/ tux@knox:~> cd D
```

5 The prompt still shows your initial input. Type the next character of the subdirectory you want to go to and press [] again.

Bash now completes the path.

6 You can now execute the command with [].

Procedure 17.7 Using Wildcards

Now suppose that your home directory contains a number of files with various file extensions. It also holds several versions of one file which you saved under different filenames `myfile1.txt`, `myfile2.txt` etc. You want to search for certain files according to their properties.

1 First, create some test files in your home directory:

1a Use the `touch` command to create several (empty) files with different file extensions, for example `.pdf`, `.xml` and `.jpg`.

You can do this consecutively (do not forget to use the Bash history function) or with only one `touch` command: simply add several filenames separated by a space.

1b Create at least two files that have the same file extension, for example `.html`.

1c To create several “versions” of one file, enter

```
touch myfile{1..5}.txt
```

This command creates five consecutively numbered files:

```
myfile1.txt,…,myfile5.txt
```

1d List the contents of your home directory. It should look similar to this:

```
-rw-r--r-- 1 tux users   0 2006-07-14 13:34
 foo.xml
-rw-r--r-- 1 tux users   0 2006-07-14 13:47
 home.html
-rw-r--r-- 1 tux users   0 2006-07-14 13:47
 index.html
-rw-r--r-- 1 tux users   0 2006-07-14 13:47
 toc.html
-rw-r--r-- 1 tux users   0 2006-07-14 13:34
 manual.pdf
-rw-r--r-- 1 tux users   0 2006-07-14 13:49
 myfile1.txt
```

```
-rw-r--r-- 1 tux users   0 2006-07-14 13:49
 myfile2.txt
-rw-r--r-- 1 tux users   0 2006-07-14 13:49
 myfile3.txt
-rw-r--r-- 1 tux users   0 2006-07-14 13:49
 myfile4.txt
-rw-r--r-- 1 tux users   0 2006-07-14 13:49
 myfile5.txt
-rw-r--r-- 1 tux users   0 2006-07-14 13:32
 tux.png
```

2 With the help of wild cards, select certain subsets of the files according to various criteria:

2a To list all files with the `.html` extension, enter

```
ls -l *.html
```

2b To list all "versions" of `myfile.txt`, enter

```
ls -l myfile?.txt
```

Note that you can only use the `?` wild card here because the numbering of the files is single-digit. As soon as you have a file named `myfile10.txt` you must to use the `*` wild card to view all versions of `myfile.txt` (or add another question mark, so your string looks like `myfile??.txt`).

2c To remove, for example, version 1-3 and version 5 of `myfile.txt`, enter

```
rm myfile[1-3,5].txt
```

2d Check the result with

```
ls -l
```

Of all `myfile.txt` versions only `myfile4.txt` should be left.

You can also combine several wild cards in one command. In the example above, `rm myfile[1-3,5].*` would lead to the same result as `rm myfile[1-3,5].txt` because there are only files with the extension `.txt` available.

---

NOTE: Using Wildcards in rm Commands

Wildcards in a `rm` command can be very useful but also dangerous: you might delete more files from your directory than intended. To see which files would be affected by the `rm`, run your wildcard string with `ls` instead of `rm` first.

---

## 17.7 Editing Texts

In order to edit files from the command line, you will need to know the vi editor. vi is a default editor which can be found on nearly every UNIX/Linux system. It can run several operating modes in which the keys you press have different functions. This does not make it very easy for beginners, but you should know at least the most basic operations with vi. There may be situations where no other editor than vi is available.

Basically, vi makes use of three operating modes:

command mode
: In this mode, vi accepts certain key combinations as commands. Simple tasks such as searching words or deleting a line can be executed.

insert mode
: In this mode, you can write normal text.

extended mode
: In this mode, also known as colon mode (as you have to enter a colon to switch to this mode), vi can execute also more complex tasks such as searching and replacing text.

In the following (very simple) example, you will learn how to open and edit a file with vi, how to save your changes and quit vi.

### 17.7.1 Example: Editing with vi

---

NOTE: Display of Keys

In the following, find several commands that you can enter in vi by just pressing keys. These appear in uppercase as on a keyboard. If you need to enter a key in uppercase, this is stated explicitly by showing a key combination including the [] key.

---

1. To create and open a new file with vi, enter

   ```
   vi textfile.txt
   ```

   By default, vi opens in command mode in which you cannot enter text.

2. Press [I] to switch to insert mode. The bottom line changes and indicates that you now can insert text.

3. Write some sentences. If you want to insert a new line, first press [] to switch back to command mode. Press [O] to insert a new line and to switch to insert mode again.

4. In the insert mode, you can edit the text with the arrow keys and with [].

5. To leave vi, press [] to switch to command mode again. Then press [:] which takes you to the extended mode. The bottom line now shows a colon.

6 To leave vi and save your changes, type `wq` (`w` for `write`; `q` for `quit`) and press []. If you want to save the file under a different name, type `w filename` and press [].

   To leave vi without saving, type `q!` instead and press [].

## 17.8 Searching for Files or Contents

Bash offers you several commands to search for files and to search for the contents of files:

`locate`
: This utility is only available if you have installed the `findutils-locate` package. With this command you can find out in which directory a specified file is located. If desired, use wild cards to specify filenames. The program is very quick, because it uses a database specifically created for the purpose (rather than searching through the entire file system). This very fact, however, also results in a major drawback: locate is unable to find any files created after the latest update of its database. The database can be generated by `root` running `updatedb`.

`find`
: With `find`, search for a file in a given directory. The first argument specifies the directory in which to start the search. The option `-name` must be followed by a search string, which may also include wild cards. Unlike `locate`, which uses a database, `find` scans the actual directory.

`grep`
: The `grep` command finds a specific search string in the specified text files. If the search string is found, the command displays the line in which `searchstring` was found, along with the filename. If desired, use wild cards to specify filenames.

### 17.8.1 Examples for Searching

The KDE and GNOME desktops store user-specific application data in hidden directories, for example `.kde` and `.gnome`.

1 To locate these directories on your computer, enter

   ```
   locate .kde
   ```

   if you have installed KDE desktop or

   ```
   locate .gnome
   ```

   if you have installed GNOME desktop.

   You will see that `locate` displays all file names in the database that contain the string `.kde` or `.gnome` anywhere. To learn how to modify this behavior refer to the man page of `locate`.

2 To search your home directory for all occurrences of filenames that contain the file extension `.txt`, use

```
find ~ -name '*.txt' -print
```

3 To search a directory (in this case, your home directory) for all occurrences of files which contain, for example, the word `music`, enter

```
grep music ~/*
```

Note that grep is case-sensitive— unless you use it with the `-i` option. With the command above you will not find any files containing `Music`.

If you want to use a search string which consists of more than one word, enclose the string in double quotation marks, for example:

```
grep "music is great" ~/*
```

## 17.9 Viewing Text Files

When searching for the contents of a file with `grep`, the output gives you the line in which the `searchstring` was found along with the filename. Often this contextual information is still not enough information to decide whether you want to open and edit this file. Bash offers you several commands to have a quick look at the contents of a text file directly in the shell, without opening an editor.

`head`
: With `head` you can view the first lines of a text file. If you do not specify the command any further, `head` shows the first 10 lines of a text file.

`tail`
: The `tail` command is the counterpart of `head`. If you use `tail` without any further options it displays the last 10 lines of a text file. This can be very useful to view log files of your system, where the most recent messages or log entries are usually found at the end of the file.

`less`
: With `less`, display the whole contents of a text file. To move up and down half a page use [] and []. Use [] to scroll down one page. [] takes you to the beginning, and [] to the end of the document. To end the viewing mode, press [Q].

`more`
: Instead of `less`, you can also use the older program `more`. It has basically the same function—however, it is less convenient because it does not allow you to scroll backwards. Use [] to move forward. When you reach the end of the document, the viewer closes automatically.

`cat`
: The `cat` command displays the contents of a file, printing the entire contents to the screen without interruption. As `cat` does not allow you to scroll it is not very useful as viewer but it is rather often used in combination with other commands.

# 17.10 Redirection and Pipes

Sometimes it would be useful if you could write the output of a command to a file for further editing or if you could combine several commands, using the output of one command as the input for the next one. The shell offers this function by means of redirection or pipes.

Normally, the standard output in the shell is your screen (or an open shell window) and the standard input is the keyboard. With the help of certain symbols you can redirect the input or the output to another object, such as a file or another command.

Redirection
: With `>` you can forward the output of a command to a file (output redirection), with `<` you can use a file as input for a command (input redirection).

Pipe
: By means of a pipe symbol `|` you can also redirect the output: with a pipe, you can combine several commands, using the output of one command as input for the next command. In contrast to the other redirection symbols > and <, the use of the pipe is not constrained to files.

## 17.10.1 Examples for Redirection and Pipe

1. To write the output of a command like `ls` to a file, enter

   ```
   ls -l > filelist.txt
   ```

   This creates a file named `filelist.txt` that contains the list of contents of your current directory as generated by the `ls` command.

   However, if a file named `filelist.txt` already exists, this command overwrites the existing file. To prevent this, use `>>` instead of >. Entering

   ```
   ls -l >> filelist.txt
   ```

   simply appends the output of the `ls` command to an already existing file named `filelist.txt`. If the file does not exist, it is created.

2. Redirections also works the other way round. Instead of using the standard input from the keyboard for a command, you can use a file as input:

   ```
   sort < filelist.txt
   ```

   This will force the `sort` command to get its input from the contents of `filelist.txt`. The result is shown on the screen. Of course, you can also write the result into another file, using a combination of redirections:

   ```
   sort  < filelist.txt > sorted_filelist.txt
   ```

3. If a command generates a lengthy output, like `ls -l` may do, it may be useful to pipe the output to a viewer like `less` to be able to scroll through the pages. To do so, enter

```
ls -l | less
```

The list of contents of the current directory is shown in `less`.

The pipe is also often used in combination with the `grep` command in order to search for a certain string in the output of another command. For example, if you want to view a list of files in a directory which are owned by the user `tux`, enter

```
ls -l | grep tux
```

## 17.11 Starting Programs and Handling Processes

As you have seen in Section 17.7, “Editing Texts” (page 201), programs can be started from the shell. Applications with a graphical user interface need the X Window System and can only be started from a terminal window within a graphical user interface. For example, if you want to open a file named `vacation.pdf` in your home directory from a terminal window in KDE or GNOME, simply run `okular ~/vacation.pdf` (or `evince ~/vacation.pdf`) to start a PDF viewer displaying your file.

When looking at the terminal window again you will realize that the command line is blocked as long as the PDF viewer is open, meaning that your prompt is not available. To change this, press [] + [Z] to suspend the process and enter `bg` to send the process to the background. Now you can still have a look at `vacation.pdf` while your prompt is available for further commands. An easier way to achieve this is by sending a process to the background directly when starting it. To do so, add an ampersand at the end of the command:

```
okular ~/vacation.pdf &
```

If you have started several background processes (also named jobs) from the same shell, the `jobs` command gives you an overview of the jobs. It also shows the job number in brackets and their status:

```
tux@linux:~> jobs
[1]   Running         okular book.opensuse.startup-xep.pdf &
[2]-  Running         okular book.opensuse.reference-xep.pdf &
[3]+  Stopped         man jobs
```

To bring a job to the foreground again, enter `fg` *`job_number`*.

Whereas `job` only shows the background processes started from a specific shell, the `ps` command (run without options) shows a list of all your processes—those you started. Find an example output below:

```
tux@linux:~> ps
PID TTY          TIME CMD
15500 pts/1    00:00:00 bash
28214 pts/1    00:00:00 okular
30187 pts/1    00:00:00 kwrite
30280 pts/1    00:00:00 ps
```

In case a program cannot be terminated in the normal way, use the `kill` command to stop the process (or processes) belonging to that program. To do so, specify the process ID (PID) shown by the output of `ps`. For example, to shut down the KWrite editor in the example above, enter

```
kill 30187
```

This sends a TERM signal that instructs the program to shut itself down.

Alternatively, if the program or process you want to terminate is a background job and is shown by the `jobs` command, you can also use the `kill` command in combination with the job number to terminate this process. When identifying the job with the job number, you must prefix the number with a percent character (`%`):

```
kill %job_number
```

If `kill` does not help—as is sometimes the case for “runaway” programs—try

```
kill -9 PID
```

This sends a KILL signal instead of a TERM signal, bringing the specified process to an end in most cases.

This section is intended to introduce the most basic set of commands for handling jobs and processes. Find an overview for system administrators in Section “Processes” (Chapter 2, System Monitoring Utilities, ↑System Analysis and Tuning Guide).

# 17.12 Important Linux Commands

This section gives insight into the most important commands. There are many more commands than listed in this chapter. Along with the individual commands, parameters are listed and, where appropriate, a typical sample application is introduced. To learn more about the various commands, use the manual pages, 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 [] and []. Move between the beginning and the end of a document with [] and []. End this viewing mode by pressing [Q]. Learn more about the `man` command itself with `man man`.

In the following overview, the individual command elements are written in different typefaces. The actual command and its mandatory options are always printed as `command option`. Specifications or parameters that are not required are placed in `[square brackets]`.

Adjust the settings to your needs. It makes no sense to write `ls file` if no file named `file` actually exists. You can usually combine several parameters, for example, by writing `ls -la` instead of `ls -l -a`.

## 17.12.1 File Commands

The following section lists the most important commands for file management. It covers everything from general file administration to the manipulation of file system ACLs.

### 17.12.1.1 File Administration

`ls [options] [files]`
If you run `ls` without any additional parameters, the program lists the contents of the current directory in short form.

`-l`
Detailed list

`-a`
Displays hidden files

`cp [options] source target`
Copies `source` to `target`.

-i
Waits for confirmation, if necessary, before an existing `target` is overwritten

-r
Copies recursively (includes subdirectories)

`mv [options] source target`
Copies `source` to `target` then deletes the original `source`.

-b
Creates a backup copy of the `source` before moving

-i
Waits for confirmation, if necessary, before an existing `targetfile` is overwritten

`rm [options] files`
Removes the specified files from the file system. Directories are not removed by `rm` unless the option `-r` is used.

`-r`
Deletes any existing subdirectories

`-i`
Waits for confirmation before deleting each file

`ln [options] source target`
Creates an internal link from `source` to `target`. Normally, such a link points directly to `source` on the same file system. However, if `ln` is executed with the `-s` option, it creates a symbolic link that only points to the directory in which `source` is located, enabling linking across file systems.

-s
Creates a symbolic link

`cd [options] [directory]`
Changes the current directory. `cd` without any parameters changes to the user's home directory.

`mkdir [options] directory`
Creates a new directory.

`rmdir [options] directory`
Deletes the specified directory if it is already empty.

`chown [options] username[:[group]] files`
Transfers ownership of a file to the user with the specified username.

`-R`
Changes files and directories in all subdirectories

`chgrp [options] groupname files`
Transfers the group ownership of a given `file` to the group with the specified group name. The file owner can change group ownership only if a member of both the current and the new group.

`chmod [options] mode files`
Changes the access permissions.

The `mode` parameter has three parts: `group`, `access`, and `access type`. `group` accepts the following characters:

u
User

g
Group

o
Others

For `access`, grant access with `+` and deny it with `-`.

The `access type` is controlled by the following options:

r
Read

w
Write

x
Execute—executing files or changing to the directory

s
Setuid bit—the application or program is started as if it were started by the owner of the file

As an alternative, a numeric code can be used. The four digits of this code are composed of the sum of the values 4, 2, and 1—the decimal result of a binary mask. The first digit sets the set user ID (SUID) (4), the set group ID (2), and the sticky (1) bits. The second digit defines the permissions of the owner of the file. The third digit defines the permissions of the group members and the last digit sets the permissions for all other users. The read permission is set with 4, the write permission with 2, and the permission for executing a file is set with 1. The owner of a file would usually receive a 6 or a 7 for executable files.

`gzip [parameters] files`

This program compresses the contents of files using complex mathematical algorithms. Files compressed in this way are given the extension `.gz` and need to be uncompressed before they can be used. To compress several files or even entire directories, use the `tar` command.

-d

Decompresses the packed gzip files so they return to their original size and can be processed normally (like the command `gunzip`)

`tar options archive files`

`tar` puts one or more files into an archive. Compression is optional. `tar` is a quite complex command with a number of options available. The most frequently used options are:

`-f`

Writes the output to a file and not to the screen as is usually the case

`-c`

Creates a new tar archive

`-r`

Adds files to an existing archive

`-t`

Outputs the contents of an archive

`-u`

Adds files, but only if they are newer than the files already contained in the archive

`-x`

Unpacks files from an archive (extraction)

`-z`

Packs the resulting archive with `gzip`

`-j`

Compresses the resulting archive with `bzip2`

`-v`

Lists files processed

The archive files created by `tar` end with `.tar`. If the tar archive was also compressed using `gzip`, the ending is `.tgz` or `.tar.gz`. If it was compressed using `bzip2`, the ending is `.tar.bz2`.

`locate patterns`
: This command is only available if you have installed the `findutils-locate` package. The `locate` command can find in which directory a specified file is located. If desired, use wild cards to specify filenames. The program is very fast, because it uses a database specifically created for the purpose (rather than searching through the entire file system). This very fact, however, also results in a major drawback: locate is unable to find any files created after the latest update of its database. The database can be generated by `root` with `updatedb`.

`updatedb [options]`
: This command performs an update of the database used by `locate`. To include files in all existing directories, run the program as `root`. It also makes sense to place it in the background by appending an ampersand (`&`), so you can immediately continue working on the same command line (`updatedb &`). This command usually runs as a daily cron job (see `cron.daily`).

`find [options]`
: With `find`, search for a file in a given directory. The first argument specifies the directory in which to start the search. The option `-name` must be followed by a search string, which may also include wild cards. Unlike `locate`, which uses a database, `find` scans the actual directory.

### 17.12.1.2 Commands to Access File Contents

`file [options] [files]`
: With `file`, detect the contents of the specified files.

  -z
  : Tries to look inside compressed files

`cat [options] files`
: The `cat` command displays the contents of a file, printing the entire contents to the screen without interruption.

  -n
  : Numbers the output on the left margin

`less [options] files`
: This command can be used to browse the contents of the specified file. Scroll half a screen page up or down with [PgUp] and [PgDn] or a full screen page down with [Space]. Jump to the beginning or end of a file using [Home] and [End]. Press [Q] to exit the program.

`grep [options] searchstring files`
: The grep command finds a specific search string in the specified files. If the search string is found, the command displays the line in which `searchstring` was found along with the filename.

`-i`
Ignores case

`-H`
Only displays the names of the relevant files, but not the text lines

`-n`
Additionally displays the numbers of the lines in which it found a hit

`-l`
Only lists the files in which `searchstring` does not occur

`diff [options] file1 file2`
The `diff` command compares the contents of any two files. The output produced by the program lists all lines that do not match. This is frequently used by programmers who need only to send their program alterations and not the entire source code.

`-q`
Only reports whether the two files differ

`-u`
Produces a “unified” diff, which makes the output more readable

### 17.12.1.3 File Systems

`mount [options] [device] mountpoint`
This command can be used to mount any data media, such as hard disks, CD-ROM drives, and other drives, to a directory of the Linux file system.

`-r`
Mount read-only

`-t filesystem`
Specify the file system, commonly `ext2` for Linux hard disks, `msdos` for MS-DOS media, `vfat` for the Windows file system, and `iso9660` for CDs

For hard disks not defined in the file `/etc/fstab`, the device type must also be specified. In this case, only `root` can mount it. If the file system needs to also be mounted by other users, enter the option `user` in the appropriate line in the `/etc/fstab` file (separated by commas) and save this change. Further information is available in the `mount(1)` man page.

`umount [options] mountpoint`
This command unmounts a mounted drive from the file system. To prevent data loss, run this command before taking a removable data medium from its drive. Normally, only `root` is allowed to run the commands `mount` and `umount`. To enable other users to run these commands, edit the `/etc/fstab` file to specify the option `user` for the relevant drive.

## 17.12.2 System Commands

The following section lists a few of the most important commands needed for retrieving system information and controlling processes and the network.

### 17.12.2.1 System Information

`df [options] [directory]`
: The `df` (disk free) command, when used without any options, displays information about the total disk space, the disk space currently in use, and the free space on all the mounted drives. If a directory is specified, the information is limited to the drive on which that directory is located.

  `-h`
  : Shows the number of occupied blocks in gigabytes, megabytes, or kilobytes—in human-readable format

  `-T`
  : Type of file system (ext2, nfs, etc.)

`du [options] [path]`
: This command, when executed without any parameters, shows the total disk space occupied by files and subdirectories in the current directory.

  `-a`
  : Displays the size of each individual file

  `-h`
  : Output in human-readable form

  `-s`
  : Displays only the calculated total size

`free [options]`
: The command `free` displays information about RAM and swap space usage, showing the total and the used amount in both categories. See Section “The free Command” (Chapter 7, Special System Features, ↑Reference) for more information.

  `-b`
  : Output in bytes

  `-k`
  : Output in kilobytes

  `-m`
  : Output in megabytes

`date [options]`
: This simple program displays the current system time. If run as `root`, it can also be used to change the system time. Details about the program are available in the date(1) man page.

### 17.12.2.2 Processes

`top [options]`

`top` provides a quick overview of the currently running processes. Press [H] to access a page that briefly explains the main options for customizing the program.

`ps [options] [process_ID]`

If run without any options, this command displays a table of all your own programs or processes—those you started. The options for this command are not preceded by hyphen.

aux
: Displays a detailed list of all processes, independent of the owner

`kill [options] process_ID`

Unfortunately, sometimes a program cannot be terminated in the normal way. In most cases, you should still be able to stop such a runaway program by executing the `kill` command, specifying the respective process ID (see `top` and `ps`). `kill` sends a TERM signal that instructs the program to shut itself down. If this does not help, the following parameter can be used:

-9
: Sends a KILL signal instead of a TERM signal, bringing the specified process to an end in almost all cases

`killall [options] processname`

This command is similar to `kill`, but uses the process name (instead of the process ID) as an argument, killing all processes with that name.

### 17.12.2.3 Network

`ping [options] hostname_or_IP address`

The `ping` command is the standard tool for testing the basic functionality of TCP/IP networks. It sends a small data packet to the destination host, requesting an immediate reply. If this works, `ping` displays a message to that effect, which indicates that the network link is basically functioning.

`-c`*`number`*
: Determines the total number of packages to send and ends after they have been dispatched (by default, there is no limitation set)

`-f`
: flood ping: sends as many data packages as possible; a popular means, reserved for `root`, to test networks

`-i`*`value`*
: Specifies the interval between two data packages in seconds (default: one second)

`host [options] hostname [server]`

The domain name system resolves domain names to IP addresses. With this tool, send queries to name servers (DNS servers).

`ssh [options] [user@]hostname [command]`
SSH is actually an Internet protocol that enables you to work on remote hosts across a network. SSH is also the name of a Linux program that uses this protocol to enable operations on remote computers.

#### 17.12.2.4 Miscellaneous

`passwd [options] [username]`
Users may change their own passwords at any time using this command. The administrator `root` can use the command to change the password of any user on the system.

`su [options] [username]`
The `su` command makes it possible to log in under a different username from a running session. Specify a username and the corresponding password. The password is not required from `root`, because `root` is authorized to assume the identity of any user. When using the command without specifying a username, you are prompted for the `root` password and change to the superuser (`root`). Use `su -` to start a login shell for a different user.

`halt [options]`
To avoid loss of data, you should always use this program to shut down your system.

`reboot [options]`
Does the same as `halt` except the system performs an immediate reboot.

`clear`
This command cleans up the visible area of the console. It has no options.

### 17.12.3 For More Information

There are many more commands than listed in this chapter. For information about other commands or more detailed information, the O'Reilly publication Linux in a Nutshell is recommended.

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

# 18.1 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.

## 18.1.1 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 commandline.

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 18.1 Bash Configuration Files for Login Shells

| File | Description |
|---|---|
| `/etc/profile` | Do not modify this file, otherwise your modifications can be destroyed during your next update! |
| `/etc/profile.local` | Use this file if you extend `/etc/profile` |
| `/etc/profile.d/` | Contains system-wide configuration files for specific programs |
| `~/.profile` | Insert user specific configuration for login shells here |

Table 18.2 Bash Configuration Files for Non-Login Shells

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

Additionally, Bash uses some more files:

Table 18.3 Special Files for Bash

| File | Description |
|---|---|
| `~/.bash_history` | Contains a list of all commands you have been typing |
| `~/.bash_logout` | Executed when logging out |

## 18.1.2 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 18.4 Overview of a Standard Directory Tree

| Directory | Contents |
|---|---|
| `/` | Root directory—the starting point of the directory tree. |
| `/bin` | 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. |
| `/boot` | Static files of the boot loader. |
| `/dev` | Files needed to access host-specific devices. |

| Directory | Contents |
| --- | --- |
| `/etc` | Host-specific system configuration files. |
| `/home` | Holds the home directories of all users who have accounts on the system. However, `root`'s home directory is not located in `/home` but in `/root`. |
| `/lib` | Essential shared libraries and kernel modules. |
| `/media` | Mount points for removable media. |
| `/mnt` | Mount point for temporarily mounting a file system. |
| `/opt` | Add-on application software packages. |
| `/root` | Home directory for the superuser `root`. |
| `/sbin` | Essential system binaries. |
| `/srv` | Data for services provided by the system. |
| `/tmp` | Temporary files. |
| `/usr` | Secondary hierarchy with read-only data. |
| `/var` | Variable data such as log files. |
| `/windows` | 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 openSUSE.

`/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 scripts that are 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. KDE users find the personal configuration data for their desktop in `.kde4` and GNOME users find it in `.gconf`.

---

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, USB sticks and digital cameras (if they use USB). `/media` generally holds any type of drive except the hard drive of your system. As soon as 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.

`/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 and establishes a secondary hierarchy in the file system. KDE4 and GNOME are also located here. `/usr` holds a number of 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/`*`packagename`* 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 A.2, “Log Files” (page 234).

`/windows`
: Only available if you have both Microsoft Windows and Linux installed on your system. Contains the Windows data available on the Windows partition of your system. Whether you can edit the data in this directory depends on the file system your Windows partition uses. If it is FAT32, you can open and edit the files in this directory. For NTFS, openSUSE also includes write access support. However, the driver for the NTFS-3g file system has limited functionality. Learn more in Section “Accessing Files on Different OS on the Same Computer” (Chapter 23, Copying and Sharing Files, ↑Reference).

## 18.2 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 18.1 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 have 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 fullfilled 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 just with `hello.sh`.

## 18.3 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 > charac-

ter. 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
```

## 18.4 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.

## 18.5 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 18.5 Useful Environment Variables

| | |
|---|---|
| `HOME` | the home directory of the current user |

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

### 18.5.1 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" "" ""
```

### 18.5.2 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
```

## 18.6 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
```

# 18.7 Working with Common Flow Constructs

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

## 18.7.1 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.

### 18.7.2 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
```

## 18.8 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

# A.1 Help and Documentation

openSUSE® 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 A.1.1, “Documentation Directory” (page 229).

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 A.1.2, “Man Pages” (page 231) and Section A.1.3, “Info Pages” (page 232).

Desktop Help Centers
: The help centers of both the KDE desktop (KDE help center) and the GNOME desktop (Help) provide 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 Novell/SUSE manuals delivered with your product.

Separate Help Packages for Some Applications
: When installing new software with YaST, the software documentation is installed automatically (in most cases) and usually 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.

## A.1.1 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, just 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.

### A.1.1.1 Novell/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 Novell/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 Novell/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 openSUSE 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/documentation/`.

### A.1.1.2 HOWTOs

If the `howto` package is installed on your system, `/usr/share/doc` also holds the `howto` subdirectory, where you find additional documentation for many tasks relating to the setup and operation of Linux software.

### A.1.1.3 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 just 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.

### A.1.2 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 [] and []. Move between the beginning and the end of a document with [] and []. 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 A.1, “Man Pages—Categories and Descriptions” (page 231) (taken from the man page for man itself).

Table A.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 `/dev`) |
| 5 | File formats and conventions (`/etc/fstab`) |
| 6 | Games |

| Number | Description |
| --- | --- |
| 7 | Miscellaneous (including macro packages and conventions), for example, man(7), groff(7) |
| 8 | System administration commands (usually only for `root`) |
| 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.

### A.1.3 Info Pages

Info pages are another important source of information on your system. Usually, they are more detailed than man pages. 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 [] to move forward and [] to move backwards. Within a node, you can also browse with [] and [] but only [] and [] will take you also to the previous or subsequent node. Press [Q] to end the viewing mode. Not every man page comes with an info page and vice versa.

### A.1.4 Online Resources

In addition to the online versions of the openSUSE 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 openSUSE check out your product-specific documentation Web page at `http://doc.opensuse.org`.

If you are searching for additional product-related information, you can also refer to the following Web sites:

openSUSE Wiki
: Detailed information about lots of different aspects of the openSUSE system can be found in our Wiki at `http://en.opensuse.org`. You may also contribute to each Wiki page and change or add new pages.

openSUSE Support Database
: The openSUSE Support Database (SDB) can be found at `http://en.opensuse.org/Portal:Support_database`. It features articles written as solutions for technical problems with openSUSE. They cover issues from installation and configuration, to workarounds for bugs and missing features.

openSUSE Forums
: There are several openSUSE forums where you can dive in on discussions about openSUSE, or get support. See http://en.opensuse.org/openSUSE:Forums_list for a list.

KDE Documentation
: Find documentation for many aspects of KDE suitable for users and administrators at http://www.kde.org/documentation/.

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's 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 is also available on openSUSE

You may also want to try general-purpose search engines. For example, use search terms `Linux CD-RW help` or `OpenOffice file conversion problem` if you have trouble with burning CDs or LibreOffice file conversion. Google™ also has a Linux-specific search engine at http://www.google.com/linux that you might find useful.

## A.2 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 Even if your situation is not precisely listed here, there may be one similar enough to offer hints to the solution of your problem.

### A.2.1 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 of which are relevant to openSUSE systems. Most log files can be viewed with YaST (Miscellaneous > Start-Up Log).

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 A.2 Log Files

| Log File | Description |
|---|---|
| `~/.xsession-errors` | Messages from the desktop applications currently running. |
| `/var/log/apparmor/` | Log files from AppArmor, see Part “Confining Privileges with AppArmor” (↑Security Guide) for detailed information. |
| `/var/log/boot.msg` | Messages from the kernel reported during the boot process. |
| `/var/log/mail.*` | Messages from the mail system. |
| `/var/log/messages` | Ongoing messages from the kernel and system log daemon (when running). |
| `/var/log/NetworkManager` | Log file from NetworkManager to collect problems with network connectivity |
| `/var/log/samba/` | Directory containing Samba server and client log messages. |
| `/var/log/SaX.log` | Hardware messages from the SaX display and KVM system. |
| `/var/log/warn` | All messages from the kernel and system log daemon with the “warning” level or higher. |
| `/var/log/wtmp` | Binary file containing user login records for the current machine session. View it with `last`. |
| `/var/log/Xorg.*.log` | Various start-up and runtime logs from the X Window system. It is useful for debugging failed X start-ups. |
| `/var/log/YaST2/` | Directory containing YaST's actions and their results. |

| Log File | Description |
| --- | --- |
| /var/log/zypper.log | Log file of zypper. |

Apart from log files, your machine also supplies you with information about the running system. See Table A.3: System Information With the /proc File System

Table A.3 System Information With the /proc File System

| File | Description |
| --- | --- |
| /proc/cpuinfo | Contains processor information, including its type, make, model, and performance. |
| /proc/dma | Shows which DMA channels are currently being used. |
| /proc/interrupts | Shows which interrupts are in use, and how many of each have been in use. |
| /proc/iomem | Displays the status of I/O (input/output) memory. |
| /proc/ioports | Shows which I/O ports are in use at the moment. |
| /proc/meminfo | Displays memory status. |
| /proc/modules | Displays the individual modules. |
| /proc/mounts | Displays devices currently mounted. |
| /proc/partitions | Shows the partitioning of all hard disks. |
| /proc/version | Displays the current version of Linux. |

Apart from the /proc file system, the Linux kernel exports information with the sysfs module, an in-memory filesystem. This module represents kernel objects, their attributes and relationships. For more information about sysfs, see the context of udev in Chapter 8, Dynamic Kernel Device Management with *udev* (↑Reference). Table A.4 contains an overview of the most common directories under /sys.

Table A.4 System Information With the /sys File System

| File | Description |
|---|---|
| `/sys/block` | Contains subdirectories for each block device discovered in the system. Generally, these are mostly disk type devices. |
| `/sys/bus` | Contains subdirectories for each physical bus type. |
| `/sys/class` | Contains subdirectories grouped together as a functional types of devices (like graphics, net, printer, etc.) |
| `/sys/device` | Contains the global device hierarchy. |

Linux comes with a number of tools for system analysis and monitoring. See Chapter 2, System Monitoring Utilities (↑System Analysis and Tuning Guide) 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.

## A.2.2 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 of the typical problems you may run into, and offers possible solutions or workarounds for these kinds of situations.

### A.2.2.1 Checking Media

If you encounter any problems using the openSUSE 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. Media problems are more probable with the media you burn yourself. Burning the media at a low speed (4x) helps avoiding problems.

### A.2.2.2 Hardware Information

Display detected hardware and technical data using Hardware > Hardware Information. Click any node of the tree for more information about a device. This module is especially useful, when submitting a support request for which you need information about your hardware.

Save the displayed hardware information to a file by clicking Save to File. Select the desired directory and filename then click Save to create the file.

### A.2.2.3 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.

Network Boot via PXE
: If a machine lacks a DVD drive, but provides a working ethernet connection, perform a completely network-based installation. See Section “Remote Installation via VNC —PXE Boot and Wake on LAN” (Chapter 2, Remote Installation, ↑Reference) and Section “Remote Installation via SSH—PXE Boot and Wake on LAN” (Chapter 2, Remote Installation, ↑Reference) for details.

External Boot Devices

Linux supports most existing DVD drives. If the system has neither a DVD drive nor a floppy disk, 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 KDE or GNOME Live CD, you can also create a “Live USB stick” to boot from. Refer to `http://en.opensuse.org/Live_USB_stick` for detailed instructions.

### A.2.2.4 Booting from Installation Media Fails

One reason why 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 the documentation provided with your motherboard, 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 a number of 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 A.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. The boot sequence looks something like `C,A` or `A,C`. In the former case, the machine first searches the hard disk (C) then the floppy drive (A) to find a bootable medium. Change the settings by pressing [PgUp] or [PgDown] until the sequence is `A,CDROM,C`.
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 A.2 Changing the Boot Sequence in a 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 A.1 US Keyboard Layout

### A.2.2.5 Fails to Boot

Some hardware types, mainly very old or very recent ones, fail to install. In many cases, this may happen because support for this type of hardware is missing in the installation

kernel, or due to 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 [] + [] + [] 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 [] 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 Chapter 1, Installation with YaST (↑Reference).

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 at all 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.

### A.2.2.6 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 A.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 Chapter 1, Installation with YaST (↑Reference).

Procedure A.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 Chapter 1, Installation with YaST (↑Reference).

Procedure A.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 [] 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 openSUSE machine and hit []:

```
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 Chapter 1, Installation with YaST (↑Reference).

---

IMPORTANT

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.

### A.2.2.7 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 hit [] 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 A.2.2.5, “Fails to Boot” (page 238). 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 [F3].

## A.2.3 Boot Problems

Boot problems are situations when your system does not boot properly (does not boot to the expected runlevel and login screen).

### A.2.3.1 Fails to Load the GRUB 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 start the Rescue System as described in Section A.2.7, “Recovering a Corrupted System” (page 255) and follow the instructions at Section “Modifying and Reinstalling the Boot Loader” (page 258).

Other reasons for the machine not booting may be BIOS-related:

BIOS Settings
: Check your BIOS for references to your hard drive. GRUB may simply not be started if the hard drive 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.

### A.2.3.2 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 runlevel or the configuration of the X Window System. To check the runlevel configuration, log in as the `root` user and check whether the machine is configured to boot into runlevel 5 (graphical desktop). A quick way to check this is to examine the contents of `/etc/inittab`, as follows:

```
tux@mercury:~> grep "id:" /etc/inittab
id:5:initdefault:
```

The returned line indicates that the machine's default runlevel (`initdefault`) is set to `5` and that it should boot to the graphical desktop. If the runlevel is set to any other number, use the YaST Runlevel Editor module to set it to `5`.

---

IMPORTANT

Do not edit the runlevel configuration manually. Otherwise SuSEconfig (run by YaST) will overwrite these changes on its next run. If you need to make manual

changes here, disable future SuSEconfig changes by setting `CHECK_INITTAB` in `/etc/sysconfig/suseconfig` to `no`.

If the runlevel is set to `5`, your desktop or X Windows 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 `/var/log/messages`. 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.

TIP: Starting X Window System Manually

One quick test: the `startx` command should force the X Window System to start with the configured defaults if the user is currently logged in on the console. If that does not work, it should log errors to the console.

## A.2.4 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 username 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.).

### A.2.4.1 Valid Username and Password Combinations Fail

This usually occurs when the system is configured to use network authentication or directory services and, for some reason, is unable to 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 why a machine appears functional but is unable to process logins correctly:

- The network is not working. For further directions on this, turn to Section A.2.5, “Network Problems” (page 248).
- DNS is not working at the moment (which prevents GNOME or KDE 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 A.2.5, “Network Problems” (page 248).
- 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 Chapter 2, Authentication with PAM (↑Security Guide).

- The home partition is encrypted. Find more information about this topic in Section A.2.4.3, “Login to Encrypted Home Partition Fails” (page 246).

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. Enter `1` at the boot prompt to make the system boot into single-user mode.
3. Enter the username and password for `root`.
4. Make all the necessary changes.
5. Boot into the full multiuser and network mode by entering `telinit 5` at the command line.

### A.2.4.2 Valid Username 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 `/var/log/messages` for error messages of the login process and of PAM.
3. Try to log in from a console (using [] + [] + [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 desktop (GNOME or KDE). For more information, refer to Section A.2.4.4, “Login Successful but GNOME Desktop Fails” (page 247) and Section A.2.4.5, “Login Successful but KDE Desktop Fails” (page 247).

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 [] + [] + [F1] and run `rm .Xauthority` as this user. This should eliminate X authentication problems for this user. Try graphical login again.

5 If graphical login still fails, do a console login with [] + [] + [F1]. Try to start an X session on another display—the first one (`:0`) is already in use:

```
startx -- :1
```

This should bring up a graphical screen and your desktop. If it does not, check the log files of the X Window System (`/var/log/Xorg.`*`displaynumber`*`.log`) or the log file for your desktop applications (`.xsession-errors` in the user's home directory) for any irregularities.

6 If the desktop could not start because of corrupt configuration files, proceed with Section A.2.4.4, “Login Successful but GNOME Desktop Fails” (page 247) or Section A.2.4.5, “Login Successful but KDE Desktop Fails” (page 247).

The following are some common reasons why network authentication for a particular user may fail on a specific machine:

- The user may have entered the wrong password.
- The username 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 hostnames, 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 username 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 `/var/log/messages` 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 [] + [] + [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 desktop (GNOME or KDE). For more information, refer to Section A.2.4.4, “Login Successful but GNOME Desktop Fails” (page 247) and Section A.2.4.5, “Login Successful but KDE Desktop Fails” (page 247).

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 [] + [] + [F1] and run `rm .Xauthority` as this user. This should eliminate X authentication problems for this user. Try graphical login again.

7 If graphical login still fails, do a console login with [] + [] + [F1]. Try to start an X session on another display—the first one (`:0`) is already in use:

```
startx -- :1
```

This should bring up a graphical screen and your desktop. If it does not, check the log files of the X Window System (`/var/log/Xorg.`*`displaynumber`*`.log`) or the log file for your desktop applications (`.xsession-errors` in the user's home directory) for any irregularities.

8 If the desktop could not start because of corrupt configuration files, proceed with Section A.2.4.4, “Login Successful but GNOME Desktop Fails” (page 247) or Section A.2.4.5, “Login Successful but KDE Desktop Fails” (page 247).

### A.2.4.3 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 have 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 [] + [] + [F1].

2 Become `root`.

3 Restart the unlocking process again with:

```
/etc/init.d/boot.crypto restart
```

4 Enter your passphrase to unlock your encrypted partition.

5 Exit the text console and switch back to the login screen with [] + [F7].

6 Log in as usual.

### A.2.4.4 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 [] + [] + [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.

### A.2.4.5 Login Successful but KDE Desktop Fails

There are several reasons why a KDE desktop would not allow users to login. Corrupted cache data can cause login problems as well as corrupt KDE desktop configuration files.

Cache data is used at desktop start-up to increase performance. If this data is corrupted, start-up is slowed down or fails entirely. Removing them forces the desktop start-up routines to start from scratch. This takes more time than a normal start-up, but data is intact after this and the user can login.

To remove the cache files of the KDE desktop, issue the following command as `root`:

```
rm -rf /tmp/kde-user /tmp/ksocket-user
```

Replace *user* with your username. Removing these two directories just removes the corrupted cache files. No real data is harmed using this procedure.

Corrupted desktop configuration files can always be replaced with the initial configuration files. If you want to recover the user's adjustments, carefully copy them back from their temporary location after the configuration has been restored, using the default configuration values.

To replace a corrupted desktop configuration with the initial configuration values, proceed as follows:

1. Switch to a text console by pressing [] + [] + [F1].
2. Log in with your username.
3. Move the KDE configuration directory and the `.skel` files to a temporary location:
   - For KDE3 use these commands:

     ```
     mv .kde  .kde-ORIG-RECOVER
     mv .skel .skel-ORIG-RECOVER
     ```

   - For KDE4 use these commands:

     ```
     mv .kde4  .kde4-ORIG-RECOVER
     mv .skel .skel-ORIG-RECOVER
     ```
4. Log out.
5. Log in again.
6. After the desktop has started successfully, copy the user's own configurations back into place:

   ```
   cp -a KDEDIR/share .kde/share
   ```

   Replace *KDEDIR* with the directory from Step 3 (page 248).

   IMPORTANT

   If the user's own adjustments caused the login to fail and continue to do so, repeat the procedure as described above, but do not copy the `.kde/share` directory.

## A.2.5 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 check list you can apply to identify the cause of any network problem encountered.

Procedure A.6 How to Identify Network Problems

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 of the 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 due to 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 will not be able to 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` or `.kde` subdirectories could not be found due to a faulty NFS server.

   Samba (File Service)
      If any application needs data stored in a directory on a faulty Samba server, it will not be able to start or function properly.

   NIS (User Management)
      If your openSUSE system relies on a faulty NIS server to provide the user data, users will not be able to log in to this machine.

   LDAP (User Management)
      If your openSUSE system relies on a faulty LDAP server to provide the user data, users will not be able to 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

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.

---

4a Use `ping` *`IP address or hostname`* (replace *`hostname`* with the hostname 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_hostname`* from another machine. If you can reach your machine from another machine, it is the server that is not running at all or not configured correctly.

If ping fails with `unknown host`, the name service is not configured correctly or the hostname used was incorrect. For further checks on this matter, refer to Step 4b (page 250). If ping still fails, either your network card is not configured correctly or your network hardware is faulty.

4b Use `host` *`hostname`* to check whether the hostname 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 hostname 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 11, The Domain Name System (↑Reference). 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.

4c 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 `ifconfig network_device` (executed as `root`) to check whether this device was properly configured. Make sure that both `inet address` and `Mask` are 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.

4d 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 `traceroute 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 at all. Use a combination of traceroute 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.

### A.2.5.1 NetworkManager Problems

If you have a problem with network connectivity, narrow it down as described in Procedure A.6, "How to Identify Network Problems" (page 249). 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:

   ```
   rcnetwork restart -o nm
   ```
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 21, Using NetworkManager (↑Reference).

## A.2.6 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. openSUSE offers dedicated YaST modules for system backup and restoration as well as a rescue system that can be used to recover a corrupted system from the outside.

### A.2.6.1 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 a exact copy of your disc. Combined with `gzip` you save some space.

Procedure A.7 Backing up and Restoring Harddisks

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 a partition to backup, 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.

### A.2.6.2 Backing Up Critical Data

System backups can be easily managed using the YaST System Backup module:

1 As `root`, start YaST and select System > System Backup.

2 Create a backup profile holding all details needed for the backup, filename of the archive file, scope, and type of the backup:

2a Select Profile Management > Add.

2b Enter a name for the archive.

2c Enter the path to the location of the backup if you want to keep a local backup. For your backup to be archived on a network server (via NFS), enter the IP address or name of the server and the directory that should hold your archive.

2d Determine the archive type and click Next.

2e Determine the backup options to use, such as whether files not belonging to any package should be backed up and whether a list of files should be displayed prior to creating the archive. Also determine whether changed files should be identified using the time-consuming MD5 mechanism.

Use Expert to enter a dialog for the backup of entire hard disk areas. Currently, this option only applies to the Ext2 file system.

2f Finally, set the search constraints to exclude certain system areas from the backup area that do not need to be backed up, such as lock files or cache files. Add, edit, or delete items until your needs are met and leave with OK.

3 Once you have finished the profile settings, you can start the backup right away with Create Backup or configure automatic backup. It is also possible to create other profiles tailored for various other purposes.

To configure automatic backup for a given profile, proceed as follows:

1 Select Automatic Backup from the Profile Management menu.

2 Select Start Backup Automatically.

3 Determine the backup frequency. Choose daily, weekly, or monthly.

4 Determine the backup start time. These settings depend on the backup frequency selected.

5 Decide whether to keep old backups and how many should be kept. To receive an automatically generated status message of the backup process, check Send Summary Mail to User root.

6 Click OK to apply your settings and have the first backup start at the time specified.

### A.2.6.3 Restoring a System Backup

Use the YaST System Restoration module to restore the system configuration from a backup. Restore the entire backup or select specific components that were corrupted and need to be reset to their old state.

1 Start YaST > System > System Restoration.

2 Enter the location of the backup file. This could be a local file, a network mounted file, or a file on a removable device, such as a floppy or a DVD. Then click Next.

The following dialog displays a summary of the archive properties, such as the filename, date of creation, type of backup, and optional comments.

3 Review the archived content by clicking Archive Content. Clicking OK returns you to the Archive Properties dialog.

4 Expert Options opens a dialog in which to fine-tune the restore process. Return to the Archive Properties dialog by clicking OK.

5 Click Next to open the view of packages to restore. Press Accept to restore all files in the archive or use the various Select All, Deselect All, and Select Files buttons to fine-tune your selection. Only use the Restore RPM Database option if the RPM database is corrupted or deleted and this file is included in the backup.

6 After you click Accept, the backup is restored. Click Finish to leave the module after the restore process is completed.

## A.2.7 Recovering a Corrupted System

There are several reasons why 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.

### A.2.7.1 Using the Rescue System

openSUSE contains a rescue system. 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 reinstall 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 website `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:

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 (including the openSUSE FTP server). 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 Section “Wake on LAN” (Chapter 2, Remote Installation, ↑Reference).

3 Enter `root` at the `Rescue:` prompt. A password is not required.

Once you have entered the rescue system, you can make use of the virtual consoles that can be reached with [] + [F1] to [] + [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, mount, init, and shutdown, and ifconfig, ip, route, and netstat 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 file `/var/log/messages`.

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.

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 rescue system provides all tools needed for a manual file system check or repair. It contains the utilities to check and repair the `btrfs`, `ext2`, `ext3`, `ext4`, `reiserfs`, `xfs`, `dosfs`, and `vfat` file systems.

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. First mount the root partition from the installed system and the device file system (change the device name to your current settings):

   ```
   mount /dev/sda6 /mnt
   mount --bind /dev /mnt/dev
   ```

2. Now you can “change root” into the new environment:

   ```
   chroot /mnt
   ```

3. Then mount `/proc` and `/sys`:

   ```
   mount /proc
   mount /sys
   ```

4. Finally, mount the remaining partitions from the installed system:

   ```
   mount -a
   ```

5. 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 exactly the same. 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 USB stick. 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 [] + [F1] to [] + [F6].

---

### Modifying and Reinstalling 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 reinstall the boot loader, proceed as follows:

1. Perform the necessary steps to access the installed system as described in Section “Accessing the Installed System” (page 257).
2. Check whether the following files are correctly configured according to the GRUB configuration principles outlined in Chapter 6, The Boot Loader GRUB (↑Reference) and apply fixes if necessary.
   - `/etc/grub.conf`
   - `/boot/grub/device.map`
   - `/boot/grub/menu.lst`
   - `/etc/sysconfig/bootloader`
3. Reinstall the boot loader using the following command sequence:

   ```
   grub --batch < /etc/grub.conf
   ```
4. Unmount the partitions, log out from the “change root” environment, and reboot the system:

   ```
   umount -a
   exit
   reboot
   ```

### 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.

---

A similar case is when you need to reinstall or update a broken driver for a device not supported by openSUSE. 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 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 openSUSE 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 “Accessing the Installed System” (page 257).

6. If using DUD, install/reinstall/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.

   6a. Identify your DVD device with `hwinfo --cdrom` and mount it with `mount /dev/sr0 /mnt`.

   6b. Navigate to the directory where your kernel files are stored on the DVD, for example `cd /mnt/suse/x86_64/`.

   6c. Install required `kernel-*`, `kernel-*-base`, and `kernel-*-extra` packages of your flavor with the `rpm -i` command.

   6d. After the installation finishes, check that a new menu entry relevant for the newly installed kernel was added to the boot loader configuration file (`/boot/grub/menu.lst` for `grub`).

7. Update configuration files and reinitialize the boot loader if needed. For more information, see Section “Modifying and Reinstalling the Boot Loader” (page 258)

8. Remove any bootable media from the system drive and reboot.

# A.3 Legal Notice

Copyright© 2006–2012 Novell, Inc. 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 Novell trademarks, see the Novell Trademark and Service Mark list http://www.novell.com/company/legal/trademarks/tmlist.html. All other third party trademarks are the property of their respective owners. A trademark symbol (®, ™ etc.) denotes a Novell trademark; an asterisk (*) denotes a third party trademark.

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

## GNU Free Documentation License

Version 1.2, November 2002

Copyright (C) 2000,2001,2002 Free Software Foundation, Inc. 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA

Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

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 noncommercially. Secondarily, 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 in order 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.

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 drawing 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.

## VERBATIM COPYING

You may copy and distribute the Document in any medium, either commercially or noncommercially, 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.

## 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.

## 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.

### 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".

### 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.

### 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.

### 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.

### 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.

### 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

To use this License in a document you have written, include a copy of the License in the document and put the following copyright and license notices just after the title page:

```
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.