openSUSE 12.3

Start-Up

Janvier 08, 2014



Start-Up

Copyright © 2006–2014 SUSE LLC and contributors. All rights reserved.

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

For SUSE or 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 SUSE or 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 SUSE LLC, its affiliates, the authors nor the translators shall be held liable for possible errors or the consequences thereof.

Table des matières

A propos de ce guide ix
1 Documentation disponible ... ix
2 Remarques ... x
3 Conventions de la documentation ... xi
4 À propos de la création de ce manuel ... xi
5 Code source ... xi
6 Remerciements ... xii

I Installation and Basics 1

1 Installation Quick Start 3
1.1 Bienvenue dans openSUSE ... 3
1.2 Configuration requise ... 3
1.3 Installation de openSUSE ... 3

2 KDE Quick Start 11
2.1 Démarrage rapide ... 11
2.2 The Standard KDE Desktop ... 11
2.3 Starting programs ... 12
2.4 Handling Media ... 14
2.5 Adapt your desktop ... 15
2.6 Application of desktop effects ... 20
2.7 Use of virtual desktops ... 21
2.8 Management of files and folders ... 22
2.9 Browsing on the Internet ... 26
2.10 Managing Passwords ... 29
2.11 E-mail and Calendar ... 30
2.12 Instant Messaging with Kopete ... 31
2.13 Starting LibreOffice ... 32
2.14 Viewing PDF Files and Other Documents ... 32
2.15 Search for data (with Strigi) ... 34

2.16 Creating a CD or DVD 34
2.17 Manage your collection of digital images 35
2.18 Managing your music collection 36
2.19 Ending Your Session 37
2.20 For More Information 38

3 GNOME Quick Start 39
3.1 Getting Started 39
3.2 The standard GNOME desktop 39
3.3 Fallback Mode 45
3.4 "Open Application" dialog box“ 46
3.5 Manage Files, Folders and Media 46
3.6 Customizing your Work Environment 49
3.7 Browsing the Internet with Firefox 51
3.8 Managing Passwords 52
3.9 E-mail and Calendar 53
3.10 Instant Messaging with Empathy 54
3.11 Starting LibreOffice 54
3.12 Viewing PDF Files and Other Documents 55
3.13 Managing Your Digital Image Collection 55
3.14 Managing Your Music and Videos 56
3.15 Ending Your Session 57
3.16 For More Information 58

4 LibreOffice Quick Start 59
4.1 Compatibility 59
4.2 LibreOffice Modules 59
4.3 Starting LibreOffice 60
4.4 LibreOffice Writer 62
4.5 LibreOffice Calc 65
4.6 Other LibreOffice Modules 67
4.7 For More Information 71

II Managing and Updating Software 73

5 Installation et Désinstallation de Logiciels 75
5.1 Définitions 75
5.2 Utilisation de l'interface KDE (Qt) 76
5.3 Using the GNOME Interface (GTK+) 82
5.4 Managing Software Repositories and Services 86
5.5 Keeping the System Up-to-date 89

6 YaST Online Update 95
6.1 The Online Update Dialog 95
6.2 Installing Patches 98
6.3 Automatic Online Update 99

7 Installing Packages From the Internet 101

8 Installing Add-On Products 103
8.1 Add-Ons 103
8.2 Binary Drivers 104

9 Managing Software with Command Line Tools 105
9.1 Using Zypper 105
9.2 RPM—the Package Manager 114

III Administration 123

10 Managing Users with YaST 125
10.1 User and Group Administration Dialog 125
10.2 Managing User Accounts 126
10.3 Additional Options for User Accounts 128
10.4 Changing Default Settings for Local Users 133
10.5 Assigning Users to Groups 133
10.6 Managing Groups 134
10.7 Changing the User Authentication Method 135

11 Changing Language and Country Settings with YaST 137
11.1 Changing the System Language 137
11.2 Changing the Country and Time Settings 140

12 YaST in Text Mode 143
12.1 Navigation in Modules 144
12.2 Restriction of Key Combinations 145
12.3 YaST Command Line Options 145

13 Setting Up Hardware Components with YaST 147
13.1 Hardware Information 147
13.2 Setting Up Sound Cards 147
13.3 Setting Up a Scanner 150

14 Printer Operation 153
14.1 The Workflow of the Printing System 154
14.2 Methods and Protocols for Connecting Printers 154
14.3 Installing the Software 155
14.4 Setting Up a Printer 155
14.5 Network Printers 159
14.6 Printing from the Command Line 162
14.7 Special Features in openSUSE 162
14.8 Troubleshooting 164

15 Installing and Configuring Fonts for the Graphical User Interface 169
15.1 Adding Fonts 169

16 Upgrading the System and System Changes 171
16.1 Upgrading the System 171
16.2 For More Information 175

IV The Bash Shell 177

17 Shell Basics 179
17.1 Starting a Shell 179

17.2 Entering Commands 180
17.3 Working with Files and Directories 183
17.4 Becoming Root 187
17.5 File Access Permissions 188
17.6 Useful Features of the Shell 192
17.7 Editing Texts 196
17.8 Searching for Files or Contents 197
17.9 Viewing Text Files 198
17.10 Redirection and Pipes 199
17.11 Starting Programs and Handling Processes 200
17.12 Important Linux Commands 201

18 Bash and Bash Scripts 211
18.1 What is « The Shell »? 211
18.2 Writing Shell Scripts 216
18.3 Redirecting Command Events 217
18.4 Using Aliases 218
18.5 Using Variables in Bash 218
18.6 Grouping And Combining Commands 220
18.7 Working with Common Flow Constructs 221
18.8 For More Information 222

A Aide et problèmes 223
A.1 Help and Documentation 223
A.2 Common Problems and Their Solutions 227
A.3 Legal Notice 254
A.4 GNU Free Documentation License 254

A propos de ce guide

Ce manuel va vous guider dans vos premiers pas avec openSUSE®. Parcourez les différentes parties de ce manuel pour apprendre comment installer, utiliser et profiter pleinement de votre système.

Installation and Basics
: Cette partie du manuel va vous guider tout au long du processus d'installation et de configuration de base de votre système. Il vous initie à YaST, l'outil central d'installation et de configuration de votre système. Apprenez comment configurer et modifier les éléments clefs de votre système et comment se connceter à Internet.

Les bases
: Cette partie s'adresse surtout aux utilisateurs qui viennent d'un autre système d'exploitation que linux. Elle vous initie à certains concepts de linux comme la gestion des utilisateurs, la strucutre des fichiers système et les permissions d'accès aux fichiers et répertoires. Une introduction au Shell est aussi fournie bien que vous n'aurez pratiquement jamais besoin d'utiliser les lignes de commandes pour interagir avec votre système à présent.

Aide et résolutions des problèmes
: Fournit un aperçu des différentes sources où trouver de l'aide et de la documentation supplémentaire pour plus d'information ou pour réaliser une tâche bien précise avec votre système. Vous trouverez également dans cette partie une compilation des problèmes et ennuis les plus fréquents et vous apprendrez comment les résoudre par vous même.

Beaucoup de chapitres de ce manuel contiennent des liens vers des ressources documentaires supplémentaires. Cela inclut aussi bien la documentation présente dans votre système que celle présente sur Internet.

For an overview of the documentation available for your product and the latest documentation updates, refer to http://doc.opensuse.org/ or to the following section.

1 Documentation disponible

Nous fournissons des versions HTML et PDF de nos livres dans différentes langues. Les manuels suivants pour utilisateurs et administrateurs sont disponibles sur ce produit :

Start-Up (p. 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 Remarques

Plusieurs voies pour vos remarques sont disponibles :

Bugs et demandes d'améliorations
: 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.

  Si Bugzilla est nouveau pour vous, vous trouverez, à coup sûr, les articles suivants intéressants :

  - `http://en.opensuse.org/openSUSE:Submitting_bug_reports`
  - `http://en.opensuse.org/openSUSE:Bug_reporting_FAQ`

Commentaires d'utilisateurs
: 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 Conventions de la documentation

Les conventions typographiques suivantes sont utilisées dans ce manuel :

- `/etc/passwd` : noms des dossiers et des fichiers
- *`placeholder`* : remplacer *`placeholder`* avec la valeur actuelle
- `PATH` : la variable d'environnement PATH
- `ls`, `--help` : commandes, options et paramètres
- `user` : utilisateurs ou groupes
- [Alt], [Alt] + [F1]: a key to press or a key combination; keys are shown in uppercase as on a keyboard
- *File*, *File* > *Save As*: menu items, buttons
- *Dancing Penguins* (Chapitre *Penguins*, ↑Un autre manuel) : Ceci est une référence à un chapitre dans un autre manuel.

4 À propos de la création de ce manuel

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 Code source

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

6 Remerciements

Avec beaucoup d'engagement, les développeurs de Linux coopèrent à échelle mondiale pour promouvoir le développement de Linux. Nous les remercions pour leurs efforts—cette distribution n'existerait pas sans eux. Nous remercions aussi Frank Zappa and Pawar. Remerciements spéciaux, bien sûr, à Linus Torvalds.

Have a lot of fun!

Votre équipe SUSE

Partie I. Installation and Basics

1 Installation Quick Start

Utilisez les procédures suivantes pour installer une nouvelle version de openSUSE® 12.3. Ce document donne un aperçu rapide de la procédure à suivre pour réaliser une installation par défaut de openSUSE.

1.1 Bienvenue dans openSUSE

Pour des instructions d'installation plus détaillées et les stratégies de déploiement, reportez-vous à la documentation de openSUSE sur http://doc.opensuse.org/.

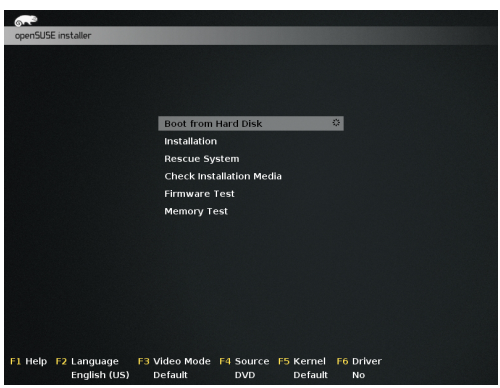
1.2 Configuration requise

- Pentium* III 500 MHz ou plus puissant (Pentium 4 2.4 GHz ou plus puissant ou n'importe quel AMD64 ou Intel* EM64T recommandé)
- 512 Mo de RAM (1 Go recommandé)
- 3 Go d'espace disque disponible (plus recommandé)
- Résolution d'affichage 800 x 600 (1024 x 768 ou plus recommandé)

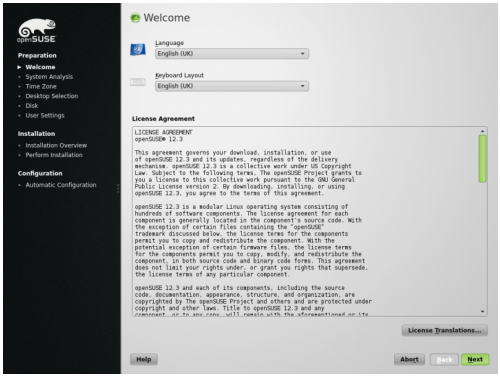
1.3 Installation de openSUSE

Utilisez ces instructions si aucun système Linux n'est installé sur votre machine, ou si vous souhaitez remplacer un système Linux existant.

1. Insérez le DVD de openSUSE dans le lecteur, puis redémarrez l'ordinateur pour lancer le programme d'installation.

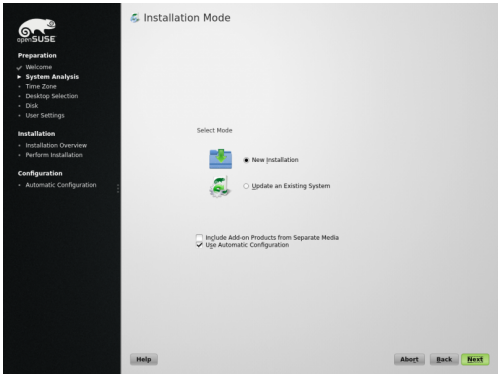


2 Sélectionnez *Installation* sur l'écran de démarrage et appuyez sur[Entrée]. Cela va charger le programme d'installation de openSUSE et démarrer l'installation en mode normal.

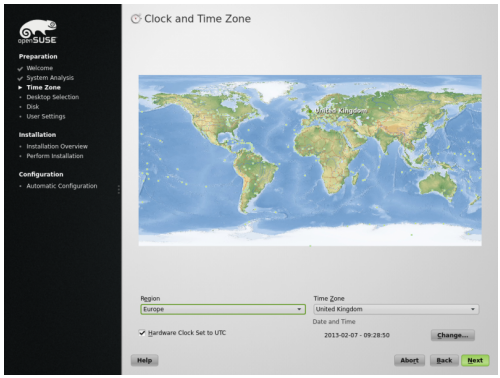


3 Sélectionnez la langue et la disposition du clavier à utiliser lors de l'installation et pour le système installé.

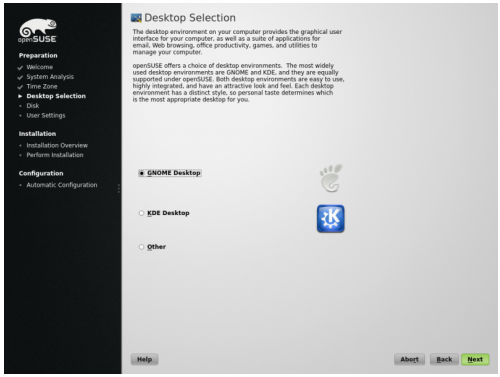
Lisez l'accord de licence et continuez avec *Suivant* si vous êtes d'accord. Si vous ne l'êtes pas, cliquez sur *Abandonner* pour interrompre l'installation.



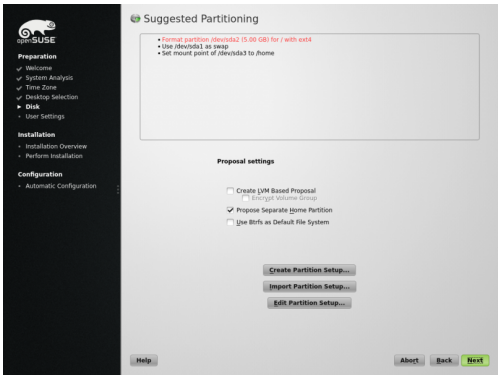
4 Sélectionnez *Nouvelle installation* et continuez avec *Suivant*.



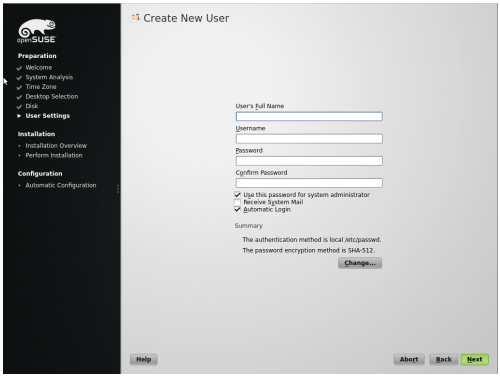
5 Utilisez soit la carte, soit les listes déroulantes *Région* et *Fuseau horaire* pour sélectionner l'heure et le fuseau horaire à utiliser sur votre système. Continuez avec *Suivant*.



6 Sélectionnez le système de bureau désiré. *KDE* et *GNOME* sont des environnements de bureau graphiques puissants semblables à Windows. Des solutions alternatives sont disponibles via *Autre*. Continuez avec *Suivant*.



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*).



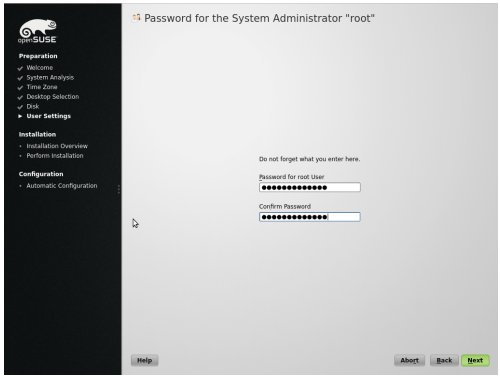
8 Type your first and last name in the *User's Full Name* field, your login name in the *Username* field, and your password in the *Password* field.

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.

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

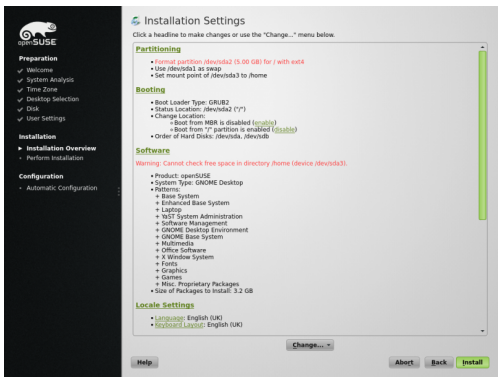
Lorsque la connexion automatique est activée, le système démarre directement sur votre bureau sans aucune authentification. Si vous stockez des données sensibles sur votre système, vous ne devriez pas activer cette option puisque l'ordinateur peut aussi être accessible par d'autres personnes.

Proceed with *Next*.



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*.

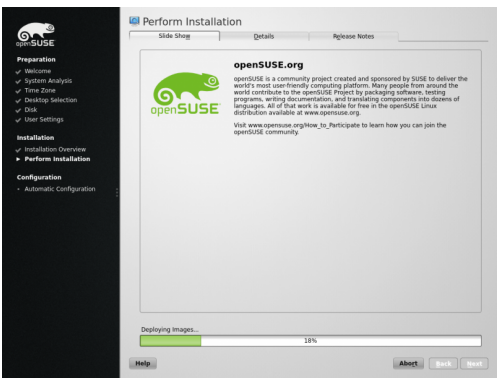


10 Utilisez l'écran des paramètres d'installation pour revoir vos choix ainsi que certains paramètres proposés automatiquement. Si nécessaire, modifiez les paramètres.

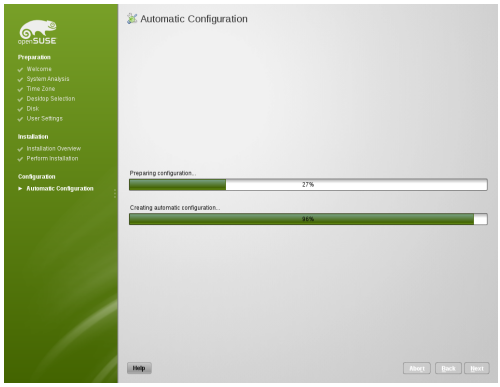
Click *Install* to proceed. Depending on your software selection you may have to accept additional license agreements by clicking *Accept*. Before the actual installation starts you will see the following screen:



11 Cliquez sur *Installer* pour démarrer l'installation.



12 L'installation dure généralement entre 15 et 30 minutes, selon les performances du système et la portée de la sélection logicielle. Pendant l'installation, vous pouvez soit voir la présentation du produit, soit les notes de versions contenant des informations de dernières minutes, soit le journal d'installation détaillé.



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 configures 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. Select your username, and type your password in the *Password* field. Press [Enter] to log in.

openSUSE® offers all tools which Linux* users require for their daily activities. The product is supplied with a user-friendly graphical environment (KDE* Desktop) which communicates with the basic Linux system in order to access files, folders, and programs and manage them. openSUSE offers an integrated suite of applications for a broad range of activities (Office, multimedia, Internet). The LibreOffice suite is also included. This enables you to edit and save files in many formats. As LibreOffice is available for different operating systems, you can use the files you have created with it on several computer platforms.

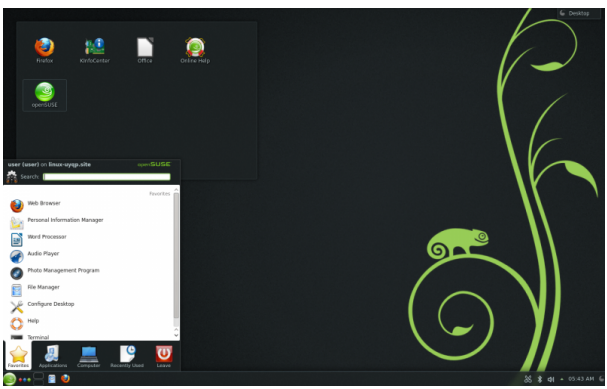
2.1 Démarrage rapide

When you start your system, you are usually prompted to enter your user name and password. If you have not installed your system yourself, ask your system administrator for the user name and password.

The KDE Desktop appears after first logging into KDE. Its appearance depends on whether openSUSE® could activate the 3D functions of your graphics card. Optics and functionality in both cases are the same, only the operation differs slightly.

2.2 The Standard KDE Desktop

When you log into KDE for the first time, the KDE desktop is displayed which contains the following basic elements:



Desktop folder: The content of the `~/Desktop` Directory is displayed as standard on the milk glass surface on the top left. Any file placed in this directory also appears in this view.

Desktop context menu: Right-click on an empty area on the desktop in order to access the Context menu which you can use to change the appearance of the desktop, add the desktop control bars or mini programs (widgets), configure desktop settings, fix mini programs in the current position (lock), lock the screen or terminate the current session.

Toolbox: Click on the icon at the very top right of the desktop with the mouse to access the menu which you can use to add or lock mini programs.

KDE control bar: As standard, the control bar on the lower edge of the KDE desktop contains the icon for the *Main Menu* on the left side and further mini programs and program icons and the icon for the control bar settings on the right side. A short description of the icon is displayed when you point the mouse pointer at an icon in the control bar.

Main menu: Click in the lower control bar on the left to open the main menu. The *Main Menu* consists of the following elements: a search function at the top and different tabs below, which offer rapid access to the key menu functions. Your registration name and the host name of the computer are also displayed in the menu.

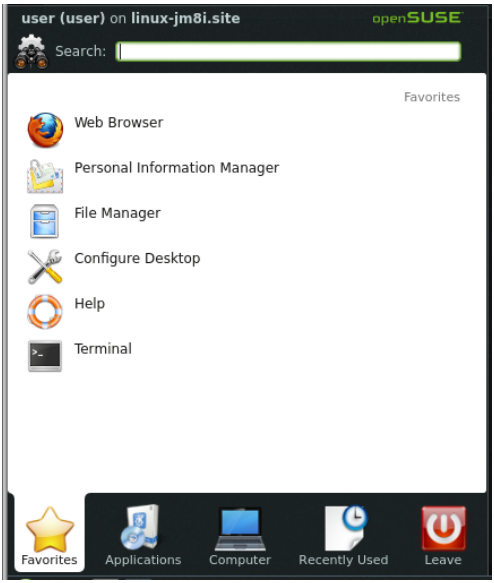
2.3 Starting programs

Start programs in the *Main Menu*, in the command line with the help of a shell or the *Execute a command* option. Programs can also be launched from the desktop or the control bar by clicking once on the program icon with the left mouse button.

2.3.1 Using the main menu

To open the *Main Menu*, click on the *Main Menu* icon in the control bar or press [Alt] + [F2].

A standard selection of pre-programmed shortcuts for quick access is displayed under *Favorites* All the applications installed in your system can be seen under *Applications*. Click on the relevant entry to navigate through the menu structure. You can return via the breadcrumb menu (the light gray text directly at the top right above the menu points). You only need to click on a previous menu point there. To jump back to Start, select *All programs*. The *Computer* and *History* tabs provide fast access to frequently used storage locations, applications, and documents. Under *Quit* you can find various options for ending the session, such as logging off, locking the screen (further access is only possible with a password), shutting down and restarting the computer. Some computers offer to put your session on RAM or a data medium into deep sleep mode.

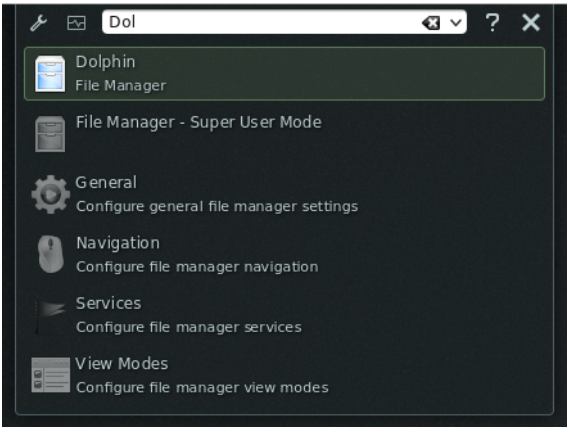


If you use a program often and do not want to access it through the menu structure every time, you can add its icon to the *Favorites* tab. To do so, right-click on any other tab in the program and select *Add to Favorites*.

2.3.2 Use the “Execute command” dialog box

KRunner is a helper application which lets you quickly start programs. It also offers a search function to search for applications or storage locations.

Press [Alt] + [F2] to open the *Execute command* dialog box. Enter a command, for example `dolphin`. Even during input, a list of applications or actions is displayed in the dialog box which match your entry. Select one of the results, or press [Entrée] to start the highlighted application or action. The command to start is often (but not always) the application name written in lower case letters.



In the *Execute command* dialog box you can use different functions: Click on the question mark icon to display a list of active functions. To edit or retrieve a list of all functions, click on the cogwheel icon. You can use so-called *web key shortcuts* which are defined in Konqueror to send search enquiries directly to a search engine without needing to open the browser beforehand. Another interesting function is the calculation of a mathematical expression such as `=2+2^4-sin(2)`. The conversion of units, such as from miles per hours to kilometers per hour, is also very useful. Simply enter the expression `60 mph in km/h` to retrieve the converted value.

2.3.3 Search for Programs

Both the *Main Menu* and the *Execute command* dialog box offer a search function which enables you to start programs quickly, even if you do not know the exact application name or command. To search for an application, start from the *Main Menu* by entering a command or a part of the application name in the *Search* box or in the input field in the *Execute command* dialog box. Every character entered restricts the search.

Select the application from the list below the input field which corresponds to your query.

2.4 Handling Media

When you place a CD or DVD into the drive, or connect a USB stick or an external hard drive, the widget opens a small window for device notification which notifies you of the process. Possible actions for the respective type of device are also suggested.



For example, if you connect a digital camera, you can download your photos with Gwenview (image viewer) or digiKam (image management program). Click on the device entry in the widget to access the available actions. To securely remove the device from the computer, click on the icon to the right of the device entry (the cable icon or the eject button).

2.5 Adapt your desktop

You can tailor the appearance and behavior of your KDE desktop to your personal requirements.

2.5.1 Fixing and approving desktop objects

Desktop elements can be fixed in the current position to prevent them moving to and from on the desktop. This does not affect the program windows, but only elements which are directly on the desktop such as the widgets or program icons. When the desktop elements are fixed, no objects can be added to the desktop, moved or removed.

Click on the Cashew icon (toolbox) in the top right of the desktop to fix or release the desktop elements and select *Lock widgets* or *Unlock widgets*.

Alternatively, right-click on an empty space on the desktop and select *Lock widgets* or *Unlock widgets*.

2.5.2 Changing individual desktop elements

In the following sections you will find some examples of how individual desktop elements can be changed.

Procédure 2.1 *Adding program icons to the desktop*

Proceed as follows to create a link to an application and to place it on the desktop or the control bar:

1. Click on the *Main Menu* and search for the desired application.

2. Right-click on the *Add to desktop* option in the displayed context menu to select it. If this menu entry is not available, your desktop elements are probably fixed. Release these first, as described in Section 2.5.1, « Fixing and approving desktop objects » (p. 15).

3. To change the position of icon symbol on the desktop, click on the icon with the left mouse button and drag it to the desired location.

If you want to delete an icon from the desktop, right-click on the program icon and select *Remove icon*.

Procédure 2.2 *Adding widgets on the desktop or on the control bar*

1. To add widgets to your desktop Right-click on an empty space on your desktop and select *Add widgets*.

   To add a widget to your control bar Right-click on an empty space on the control bar and select *Control bar Settings* > *Add widgets*.

   If these menu entries are not available, your desktop elements are probably fixed. In this case, first right-click on the control bar and select *Unlock widgets*.

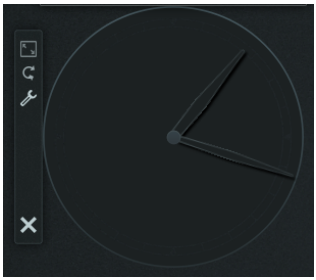
2. Select a widget from the displayed list. Search through the available widgets with the help of the scroll bar. To restrict the selection of widgets either input a search term in the input field or select a category.

3. To display the selected widget on the desktop or on the control bar, click it with the left mouse button, keep the button depressed and drag the symbol to its envisaged position. Then release the left mouse button again. The mouse pointer turns into a prohibited symbol in locations where it is impossible to place the widget.

4. To remove a widget from the desktop or the control bar, right-click on the widget and select *Remove...*.

Procédure 2.3 *Configuring widgets*

1. Hold the mouse pointer over a widget until a translucent frame appears around the widget which displays a row of icons:



If the frame does not appear, your widgets are probably fixed. Release these first, as described in Section 2.5.1, « Fixing and approving desktop objects » (p. 15).

2 To change the size of the widget, click on the ruler symbol in the frame with the left mouse button and keep the mouse button depressed while you slowly move the pointer over the desktop.

3 To rotate the widget in one direction, click on the curved arrow symbol in the frame with the left mouse button and keep the mouse button depressed while you drag the mouse pointer in a circular motion over the desktop. There is a locking limit on the horizontal and vertical positions to which you can « affix » the widget if necessary. Of course, you can also place it in any appropriate position.

4 To change the content, the settings or the properties of a widget, click on the cogwheel icon in the frame with the left mouse button. Determine the options as desired in the configuration dialog box which is now displayed.

panels are available to gain more space for your own programs.

Procédure 2.4 *Adding and removing panels*

1 Right-click on an empty space on the desktop, select *Add panel*, and a panel type from the list.

If these menu entries are not available, your desktop elements are probably fixed. In this case, first right-click and select *Unlock widgets*.

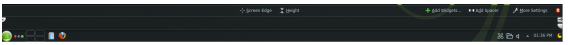
2 Click on the Cashew icon at the end of the new panel to access the configuration options and adapt the panel as described in Procédure 2.5, « Configuring panels » (p. 18).

3 To delete a panel, right-click on an empty space in the panel for deletion and select *panel Settings* > *Remove "panel"*. At least one panel must remain on the desktop in openSUSE.

Procédure 2.5 *Configuring panels*

Applications and widgets can be added in the main panel or other panels. panel elements and other panels can be shifted to different positions or removed if necessary.

1 In order to access options for the current panel, right-click on an empty space in the panel and select *Settings for panel > panel settings*. It is quicker if you click on the cashew icon at the very right of the bar. In any case, the view of the panel changes and the configuration options are displayed.



If the menu entry and the icon is not available, your desktop elements are probably fixed. To unlock, first right-click on the panel and select *Unlock widgets*.

2 In order to move objects in the panel to another position in the panel, point to the desired panel object with the mouse cursor (e.g. program icon, desktop toggle, task manager, system section for panel). The mouse pointer takes the form of a cross. Click on the object and drag the element to the desired position in the panel. Click again to fix it in this position.

3 To increase the distance between two widgets, use *Add spacers*. A spacer widget is displayed on the right hand side. Drag it to the preferred location and leave it there.

4 To change the width of the panel, click on the small arrow icon on the left and right hand side of the settings panel. With the arrow on the left hand side you can determine the position of the panel and with the arrows on the right hand side you can determine the maximized and minimized size.

5 To change the height of the panel, left-click on the *Height* button and drag to the desired final position.

6 With the help of the *Screen edge* button, position the panel on one side of the screen. Drag the panel into the desired position using drag and drop.

7 If you need additional widgets in your panel, you can add them using the *Add widgets* button as described in Procédure 2.2, « Adding widgets on the desktop or on the control bar » (p. 16).

8 To remove program icons or widgets from the panel, right-click on the object in question and select the menu entry to remove the icon or widget.

The icons in the system section of the panel normally belong to applications which are executed in the background. You can therefore only remove these icons once you have closed these applications (click on the icon and select *End*).

9 Further configuration options (e.g. orientation) and further functions (e.g. automatic masking) can be found under *Advanced settings*.

10 When all panel settings meet your requirements, close the panel settings with the red button on the right hand side.

Procédure 2.6 *Changing the Desktop Background*

You can set the desktop background in such a way that images, slide shows, samples or colors are displayed. Even a world map or the weather forecast can be used as a desktop background.

1 Right-click on an empty space on the desktop and select *Default desktop settings*. A configuration window is opened.

2 In the *View* section select a type from the *Background image* dropdown list. Further configuration options are dependent on the object you have selected for the background.

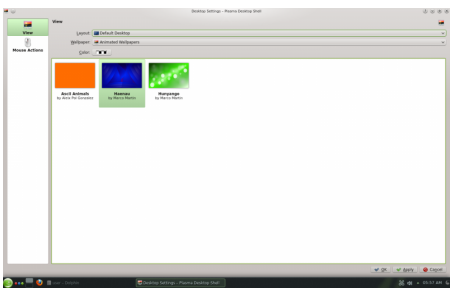
3 In order to use an image as a background, proceed as follows:

3a Choose the *Image* option under *Background image*.

3b Define the *Positioning* of the image.

3c Use one of the available background images by selecting the desired background in the list, or click on *Download new background images* to obtain further background images from the Internet.

3d To use a user-defined image, click on *Open* and select an image file from the file system.



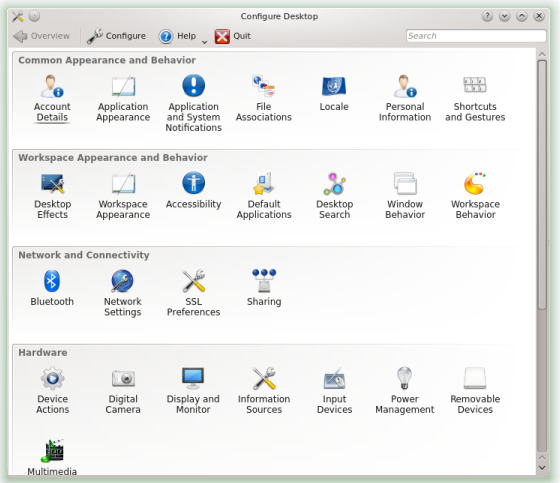
4 Determine the other options in the configuration dialog box according to your wishes and click on *OK* to save your changes and close the configuration dialog box.

2.5.3 Modify personal settings

Apart from changing individual desktop elements, you can adapt your desktop with KDE to your precise requirements. Options for adapting the entire appearance and behavior of your desktop can be found in the *Configure desktop* configuration dialog box.

Launch *Configure desktop* in the *Main Menu* by selecting *Favorites* > *Configure desktop*.

The settings are sub-divided into different categories. Hover the mouse on an icon to see further details. You need system administrator rights for some tasks in Configure desktop (`root`).



Change the settings as desired. Changes only become effective when you click on *Apply*. Click on *Restore* to discard changes in the last view which have not yet been applied. In order to restore all elements in the last view to standard values, click on *Default settings*.

To return to the initial display (with all categories), click in the top left of the dialog box on *Overview*. You can also input search terms at the top of the window (for example *Screensaver*) to find the category which contains options that match the search text. Each character you enter in the *Search* field further restricts the search. The icon of the category found is highlighted in a different color.

For a more simple overview of all categories, you can change to a tree view by clicking on *Create* and selecting a different *View design*.

2.6 Application of desktop effects

If your computer supports a 3D graphics card, you can use graphic effects, such as converting the desktop to a rotating 3D dice, activating translucent or transparent windows or maximizing and minimizing the desktop screen. You can also use other

window effects such as Shadow, Blending in and Remodeling. If you do not have a 3D graphics card, some effects may be disabled or can slow down your system. If your system is too slow, disable the effects for the current session by pressing [Alt] + [Maj] + [F12].

Configure the effects of your desktop as follows:

1. Start the *Configure desktop* as described in Section 2.5.3, « Modify personal settings » (p. 19).
2. Select *Desktop effects* from the *Desktop appearance and behavior* category, and select *Activate desktop effects upon system start*. If your graphics card does not support the 3D function, you are notified accordingly.
3. Optional: Change some options under *General settings* on the *Advanced* tab.
4. Select the effects to be activated on the *All effects* tab. When you click on the information icon, a short explanation on the respective effect is superimposed. If an effect is activated, you can obtain access to further configuration options with the cogwheel icon (right), for example to assign keyboard shortcuts for the effect.
5. With *Apply* you can activate the effects permanently.

2.7 Use of virtual desktops

The desktop environment enables 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. For example, you can use a desktop for e-mails and the calendar and another desktop for text processing or graphic applications.

Procédure 2.7 *Moving an application to another virtual desktop*

You can display a currently executed application on one or all virtual desktops or move them to other desktops.

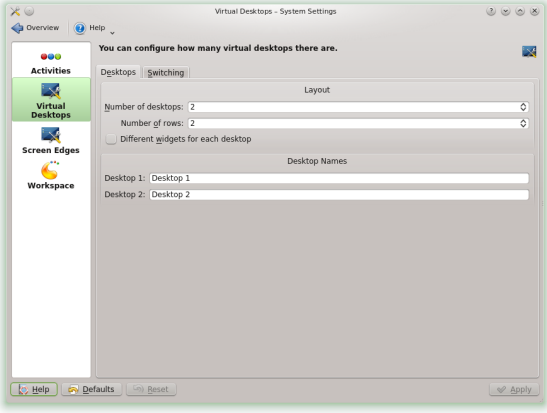
1. Open the application.
2. Right-click on the title bar of the application.
3. Click on *On Desktop*.
4. Select the desktop where the application should be placed.
5. Click on the desired desktop in the desktop toggle switch in the control bar in order to change to a certain desktop. Change to the next or previous desktop by placing the mouse pointer in an empty space on the desktop and moving the mouse wheel.

Procédure 2.8 *Configuring the virtual desktop*

You can increase the number of virtual desktops, change standard names, assign activities, configure effect animations and assign keyboard shortcuts to switch virtual desktops.

1 To add additional desktops, right-click on the desktop toggle switch in the control bar and select *Add virtual desktop*. A new desktop is displayed in the desktop toggle switch.

2 If configuration options are available, right-click on the desktop toggle switch and select *Desktop toggle switch settings*.



3 You can configure the appearance of the desktop toggle switch in the control bar with the options in the *General* category.

4 Desktop names, key combinations to change between desktops, and the switchover type can be configured in the *Virtual Desktops* category.

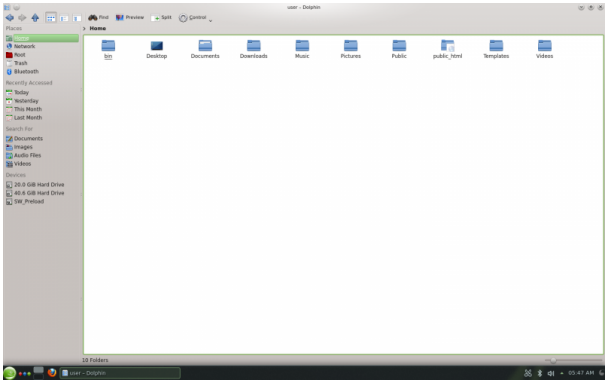
5 Determine the desired options and click on *OK*. The changes are adopted and the configuration dialog box is closed.

2.8 Management of files and folders

In KDE 4, Konqueror has been replaced by Dolphin as a standard file manager. Konqueror will continue to be used as a web browser (information regarding this can be found in Section 2.9, « Browsing on the Internet » (p. 26)). Start Dolphin in the main menu by selecting *Favorites* > *File manager* or pressing [Alt] + [F2] and entering `dolphin`.

2.8.1 Dolphin main window

The Dolphin main window consists of the following elements:



Menu bar: Call the main menu up via the cogwheel icon. All possible functions are ready there, such as copying, moving or deletion of files, changing view, starting of further tools, changing settings and calling up help.

Symbol bar: The symbol bar accelerates the navigation and changing of the display. The relevant functions can also be called up via the main menu. If the mouse pointer is positioned over an icon for a moment, a short description of the icon is displayed.

Address bar: The address bar displays the path to the current directory. It is available in two versions: one displays the path to the current directory with buttons for each superordinate folder. Click on a directory name in this view in order to change to this directory. The second version of the address bar displays the path to the current directory as a text chain which can be edited.

Fields: The *Places* sidebar is displayed in Dolphin as standard. It enables rapid access to frequently used places such as your home directory, the root directory of the file system (/), the Recycle Bin, and removable media. You can add further different control bars in the main menu under *View* > *Side bars* .

Display field (work area): The content of the selected directory or the selected file is displayed in the display field. Dolphin shows the content of your home directory as standard at the start. By clicking on a folder in Dolphin an action is started directly as standard: Dolphin loads the file into an application or opens the folder for further processing.

Status bar: Shows the file type and the size of the currently selected object and enables maximization and minimization of the icons or entries in the display field via the slider.

2.8.2 Management of folders and files in Dolphin

To perform actions such as copying, moving, creating or deleting files, you require the relevant rights for the respective folders and files.

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

1. To select one or several files and folders in Dolphin move the mouse pointer over the file or folder, but do not click on it. A green cross appears on the left upper edge. The object is selected when you click on it. To clear the selection, click on the red minus sign which is displayed when the object is selected.

   Alternatively, press [Ctrl] and click on the objects which should be selected or unselected.

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

3. Navigate to the target folder where you wish to insert the object.

4. To create a new folder at the current location, right-click in the display field and select *Create* > *Folder* or press [F10]. Enter a folder name in the new window and press [Entrée].

5. In order to insert the copied or cut object in Étape 2 (p. 24), right-click on the target folder in the main display field and select *Insert*. The object is copied or moved into this folder.

6. To delete a file or a folder, right-click in the main display field on the object and select *Send to Recycle Bin* in the context menu. The object is moved to the bin. You can retrieve it from there or permanently delete the object if necessary.

In order to quickly filter for certain file names in the current directory, press [Ctrl] + [I] to add the *Filter* input field at the bottom of the Dolphin main window. Enter any part of the file name for which you are searching to see all files in the current directory which contain the chain of characters searched for. Press [Ctrl] + [F] to use the desktop search for detailed and advanced search. You can find further information on this in Section 2.15, « Search for data (with Strigi) » (p. 34).

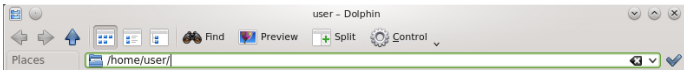
2.8.3 Configuration of Dolphin

Dolphin offers many options for changing the view and the options as a whole to suit your preferences.

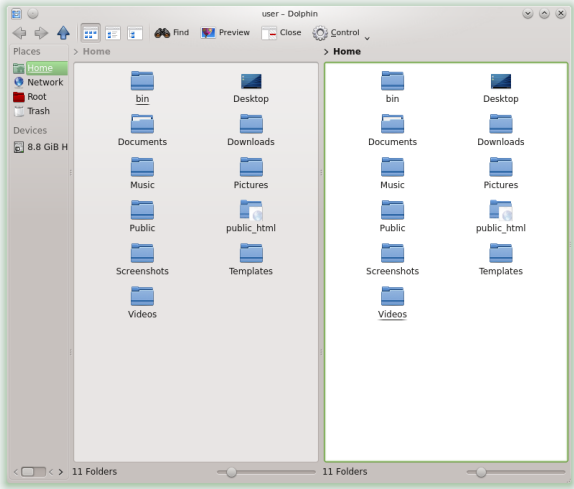
Procédure 2.9 *Changing the view*

1. To change from standard view to the editable version of the address bar, press [F6]. Enter a path to a directory. After inputting an address, press [Entrée] or click the green hook on the right adjacent to the input field.

   To delete the content in the address bar click on the right on the black X icon. To change back to the standard view, press [Ctrl] + [L].



2 To change the view of the currently displayed folder click on one of the relevant icons in the icon bar (fourth, fifth and sixth from the left). Dolphin retains the selected view for each folder. Press [F3] to display the content of the current folder in two separate columns. You can now navigate to a different directory in each column and easily drag or position objects or compare the contents of directories. Press [F3] again to close the split pane.



3 If you want Dolphin to display hidden files, click on the cogwheel icon to open the main menu and choose *Show hidden files*. (If you have enabled the menu bar, instead use *View* > *Show hidden files*.)

4 To display further details on the files (such as access rights or ownership), select *Additional Information* in the main menu behind the cogwheel icon and activate the relevant options.

5 To add further directories to the *Places* sidebar, drag a folder from the desktop into the *Places* sidebar and position it there Right-click on the context menu and use it to hide, edit, or remove entries from *Places*.

6 To add further sidebars to the main window, select *Sidebars* behind the cogwheel icon in the main menu and then further sidebars such as *Folders* or *Terminals*.

Separate the sidebars from Dolphin's main window by clicking the name of the sidebar and then dragging it to another location on the desktop. If the sidebar has no title it is locked. In such a case, right-click in a free area on the sidebar and then select *Unlock sidebars*. to integrate the sidebar into Dolphin's main window again, double-click on the name of the sidebar again.

Procédure 2.10 *Changing the overall behavior of Dolphin*

To change the behavior or the view of Dolphin as a whole, select *Configure dolphin* in the main menu behind the Cogwheel icon and view the options offered in Dolphin's configuration catalog.

1. If a certain view mode should apply to all folders, click on *General* in the left sidebar and activate *Use common properties for all folders*. Click on *Views* in the left sidebar and adapt the options for the different views. Click on *Apply* to save the changes.

2. If Dolphin displays another directory on startup, or if you want to use the editable address bar constantly, click on *Startup* in the left sidebar and change the relevant options.

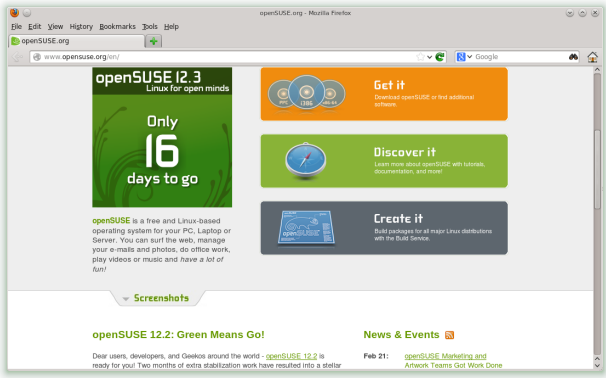
3. To configure the entries and services displayed in Dolphin's context menu, click on *General*. Activate the entries to be displayed on the *Context menu* tab. Click on *Services* in the left sidebar and activate or disable the relevant services to be displayed in the context menu.

4. Click on *OK* to accept the changes and close the Dolphin configuration dialog box.

2.9 Browsing on the Internet

Apart from Konqueror (the standard KDE web browser) openSUSE also contains the Firefox web browser. To start Konqueror or Firefox, press [Alt] + [F2] and enter `konqueror` or `firefox`.

Both browsers offer support for modern web standards, the use of tabs, a pop-up blocker, and download and image management. Simple access to different search engines helps you in searching for required information.



Enter the URL in the address field to start browsing. Press [Ctrl] + [T] to open a new, empty tab. To open a link in a new tab, click on the link with your middle mouse button. When you right-click on the tab, a menu with tab options is superimposed. This menu allows you to create new tabs, reload the content of a certain tab or all available tabs and close tabs. You can also change the sequence of the tabs by dragging it to the desired position and leaving it there.

2.9.1 Finding Information

Firefox offers two different types of search options: You can search for information on the web or on the current webpage using certain terms.

Procédure 2.11 *Searching the web*

1. To start a search on the web, click on the left icon on the right of the search bar (adjacent to the places bar) to open a list of search engines.

2. Select the desired search engine and enter your search terms into the input field.

3. Press [Entrée] to start the search process.

4. If you want to search the current webpage with Firefox or Konqueror, press [Ctrl] + [F] to display a search bar at the bottom of the window. Enter your search term there and start the search with the buttons below the search bar in different directors or configure the search options, for example *Upper/Lower case*.

Konqueror offers predefined web key combinations to quickly search the web. To search for `yast`, for example, on the web using the Google search engine enter `gg:yast` in Konqueror's address bar (or the *Execute command* dialog box) and press [Entrée].

You can also define individual Web key combinations in Konqueror and Firefox as described in Section 2.9.3, « Configuring Preferences » (p. 28).

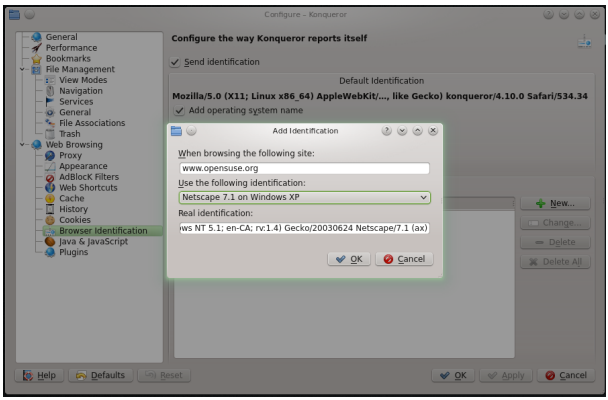
2.9.2 Downloading Files from the Web

When you download a file with Firefox or Konqueror, a dialog box is opened in which you are requested whether you want to save the file or open it with a certain application. Also specify whether the selected action should apply to all files of this type. Firefox saves the file in the folder configured in the Firefox settings as standard and shows the completed downloads in the *Downloads* dialog box.

To open the downloaded files directly from there, right-click on *Open* to select the option. To clear the history of downloaded files, click on *Empty list*.

2.9.3 Configuring Preferences

Changing the settings in Konqueror or Firefox. Access the configuration dialog box in Konqueror by selecting *Settings* > *Configure Konqueror*. To change the behavior of Konqueror as a web browser, select one of the sub-categories in *Surfing the Internet* in the left navigation area to display the options for the relevant category. For example, you can « hide » the true identity of Konqueror and identify Konqueror as another browser (such as, Internet Explorer*) for certain websites you visit. To do so, select *Browser identification* and click on *New* in order to add a new, page-specific identification.



Perhaps you would also like to configure the web key shortcut you can use with Konqueror (and the *Execute command* dialog box). Click on *Web shortcut* in the configuration dialog box of Konqueror to display the currently defined key shortcut. Click on *New* to define new key shortcuts. Press *OK* to close the configuration dialog box of Konqueror and apply the changes.

Select *Edit* > *Preferences* in Firefox to open the *Firefox settings*. Click on the symbols in the upper display field in order to access the options for the relevant category. For

example, change the standard download folder on the *General* page or change the blocking of pop-up windows on the *Content* page. Click on *Close* to accept the changes.

Navigate to the website with the desired search field to configure a web key combination for a search in Firefox. Right-click on the search field and select *Add keyword for this search...*. Enter a *Keyword* and close the dialog box by pressing *OK*.

2.10 Managing Passwords

When you input a password in a KDE application for the first time (for example in KMail or in Konqueror), you are asked if you want to save the password. If you click on *Yes*, the KWallet assistant starts as standard. KWallet is a password management tool which can collect all passwords and save them in one or more encrypted files (so-called « Wallets »).



To activate the KWallet, select *Basic set-up*, and click on *Advanced*. Select *Yes, I want to use the KDE digital wallet for my personal information*. Then assign a master password to the standard wallet (`kdewallet`), which is created in the next step.

AVERTISSEMENT : Protect your password for KWallet

If you forget your master password for a wallet, this password cannot be recreated. Also, everyone who knows your password can look inside your wallet.

Click on *Finish* to close the assistant. If you originally started the KWallet assistant via a different application such as Konqueror or KMail, you are requested in KWallet to re-enter the password so that the password can be saved in the newly created wallet for this application. After the first configuration you can open KWallet at any time to display, search for, delete or create entries. Entering passwords manually is not usually necessary. KDE recognizes when a resource needs to be authenticated and KWallet starts automatically and asks you to enter your KWallet password. However, you can

also add additional data if you wish. To configure the KWallet options, press [Alt] + [F2] and enter `kwalletmanager`. The *Service program for digital wallets* is opened. If the window does not appear, take a look at the system section of the control bar (on the left adjacent to the clock). If necessary, expand all hidden applications via the small triangle. Then click on the *Service program for digital wallets* or on the relevant icon. The window of the application should then open. Select *Settings* > *Set up digital wallet*.

While KWallet is set up in such a way that passwords for different KDE applications are managed centrally, Firefox also offers the option of saving data on a website when a user name and password are entered. When you accept this, by clicking on *Save* the password is saved on your hard drive in an encrypted format. Firefox fills out the registration errors automatically when this website is next accessed.

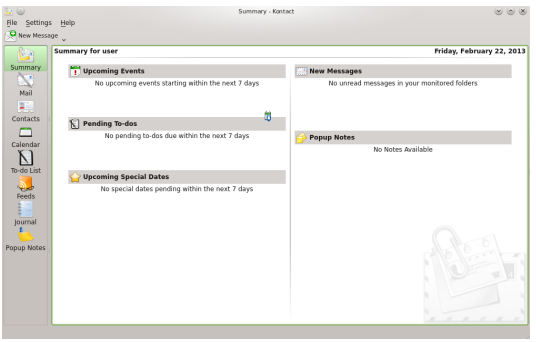
To verify or manage your passwords in Firefox, click on *Edit* > *Preferences* > *Security* > *Saved Passwords...*.

2.11 E-mail and Calendar

You can use Kontact as your personal information management (PIM) tool to read and manage your e-mails and events. Kontact combines KDE applications such as KMail, KOrganizer and KAddressBook in a single interface. This facilitates access to e-mails, calendar, address book and other PIM functions. Kontact can also manage several e-mail accounts.

2.11.1 First time start-up of Kontact

To start Kontact press [Alt] + [F2] and enter `Kontact`. To access one of the components (e.g. KMail), click on the relevant icon on the left of the sidebar.



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

1 Select *Settings* > *Configure Kontact*.

2 Click on *Identities* in the left sidebar under *E-mail* and select the identity KMail has automatically created for you. *Change* this identity by entering your full name and e-mail address and confirm your changes with *OK*.

3 Click in the left sidebar under *E-Mail* on *Accesses* and select the *Add* option on the *Receiving* and/or *Sending* of e-mails tab to create at least one account. (Optionally, you can configure several e-mail accounts.)

4 Select the type of access. To retrieve e-mails, this is usually *IMAP* or *POP3*. To find out the right values to use, please consult the help pages of your e-mail provider. Highlight the relevant entry in the list and click *OK*. KWallet reports now, to whom you name the (master) password for your wallet. Then fill in the form which appears. Dependent on the account, options must also be determined for encryption or authentication. If you are unsure which settings or objects you should search for, get in touch with your Internet Service Provider or your system administrator.

5 Click on *OK*. Set-up is confirmed and the dialog box is closed.

To write a new e-mail in the main window of Kontact, click on the *Mail* icon and press [Ctrl] + [N] to open the E-mail Editor. When you have finished your e-mail, click on *Send* to send it straightaway. If you have configured several e-mail accounts, hold down the *Send* button and select the account to send the e-mail from.

2.12 Instant Messaging with Kopete

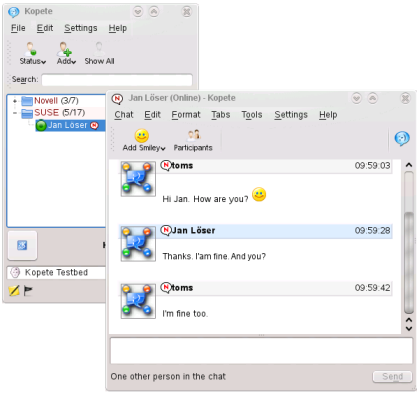
Kopete is an online messenger application which enables you to chat with others over the Internet. Kopete supports many messenger protocols, such as AOL* Instant Messenger (AIM), Gadu-Gadu, GroupWise® Messenger, ICQ, Jabber*, SMS, and Yahoo!*. To use Instant Messaging (IM), you need to register with an IM services provider and then enter this data Kopete accordingly.

To start Kopete, press [Alt] + [F2] and enter `kopete`. When you start the application for the first time, select *Settings* > *Configure*. Add an *Account* here in the *Accounts area*. Kopete takes you through the set-up steps, such as selecting the news service, entering the necessary account information (for example, user name/user ID, server name, port), inputting of registration data (e.g. passwords). If you have not yet registered with the provider for the selected news service, click on *Register New Account*. Enter your user data into the browser window which is now opened to register. Change back to Kopete and enter the data received upon registration. Close the configuration of your account by clicking on *Finish*.

If you want to go online after configuring your account, you can now add contacts in the main window of Kopete. When the status `Offline` is displayed in the Kopete window (in the status bar), first click on *File* > *Set status* > *Online* and enter your password as soon as you are requested. When connected, click on *File* > *Add contact* and select the Kopete account for which contacts should be added. Enter the contact data or search for a contact and click on *OK*.

To display all contacts, even if these are currently offline, select *Settings* > *Show Offline Users*.

When you want to start a chat, click on the desired contact and enter your message into the lower section of the chat window. Press [Entrée] to send the message. The upper part of the window displays the messages you have sent and received.



2.13 Starting LibreOffice

The office suite LibreOffice offers a comprehensive set of office tools, which includes a text processor, a spreadsheet application, a slideshow application, a vector graphics program and a database application and more. As LibreOffice is available for a range of operating systems, you can seamlessly use files created with it on different platforms.

Press [Alt] + [F2] to start and enter `libreoffice`. When you want to create a new document, select *File* > *New* and then select the desired document type. To open an existing document, click *Open* and select the relevant file from the file system.

You can find further information in LibreOffice Quick Start.

2.14 Viewing PDF Files and Other Documents

Documents which need to be approved or printed on several platforms can be saved as PDF (Portable Document Format) files, for example with the LibreOffice-Suite. Display these documents with Okular, KDE's standard document display.

2.14.1 Using the Okular document display

Apart from PDF files, you can display a multitude of file formats with Okular, such as PostScript, different image formats, the OpenDocument Format (ODF), some eBook formats (ePub) and even Microsoft* Compiled HTML Help (CHM). Okular also supports bookmarks, comments, formula fields and multimedia contents as well as page rotation.

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

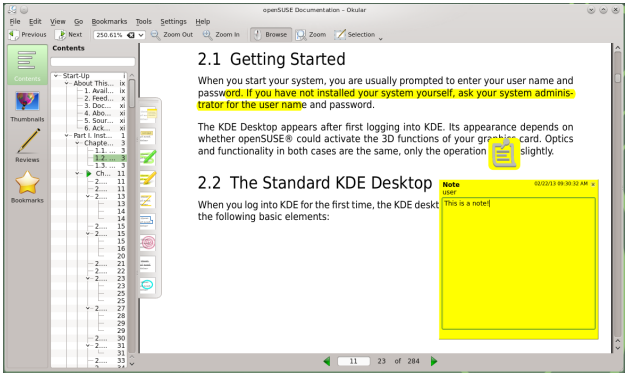
To open a document select *File* > *Open* and select the relevant file from the file system. Navigate through the document with the up or down navigation icons in the window. According to which icon you click on the left side of the navigation area, the sidebar either shows the *Content*, *Preview images* of all pages, the *Reviews* or your *Bookmarks* for this file. To filter for text which is listed in the sidebar or in the entire document, enter a string of characters into the input field at the top of the sidebar. If you want to select and copy a text or images from the file in Okular, click on the *Select* icon in the toolbar and select one of the options from the context menu. Click on the *Browse* icon to return to browsing the document.

Procédure 2.12 *Working with bookmarks and comments*

With Okular you can verify a document by emphasizing certain passages of text or adding comments or bookmarks which Okular then adds to the file in the form of metadata. Note that the added comments and identifiers cannot be saved in the document. Consequently, it is not possible to print them or forward them to other users.

1 To add a bookmark to a page click on the relevant page in the sidebar or in the main display field and click on [Ctrl] + [B]. The page is added to the list of *Bookmarks* in the left sidebar. Right-click on the bookmark entry to access a context menu in which you can change to this bookmark, rename it or remove it.

2 To create a comment for a page, press [F6] and select one of the comments tools from the toolbar now displayed. The comment is added to the list of *Reviews* and identified with the sign-in name of the user who created it. Use the symbols at the bottom of the sidebar in order to group the comments by page or author or to display only the comments for the current page.



3 To open a pop-up note and add a text for a comment (or to remove a comment), right-click on the comment entry in the list of *Reviews* and select the relevant

menu entry. The comments or identifiers added are automatically added to the file and you do not need to save these.

2.14.2 Using Acrobat Reader

If Acrobat Reader is not installed as standard, install the `acroread` package with YaST. To start Acrobat Reader, press [Alt] + [F2] and enter `acroread`. Click on *File* > *Open*, search for the desired PDF file and click on *Open*, to display the file.

2.15 Search for data (with Strigi)

The Strigi desktop search not only searches for files, but also for metadata such as tags, evaluations and even source URLs for downloads. To use desktop search, the `nepomuk` service must be executed. To activate this service, open Configure Desktop, switch to *Desktop search*, tick all services on the *Basic Settings* tab and terminate the dialog box by clicking on *Apply* and *OK*.

After activation, you can use KRunner ([Alt] + [F2]) to search for files by inputting a keyword. An advanced search interface is available in the Dolphin file manager. Press [Ctrl] + [F] in Dolphin to activate the desktop search. You can filter the search results by *File name*, *Content* or place.

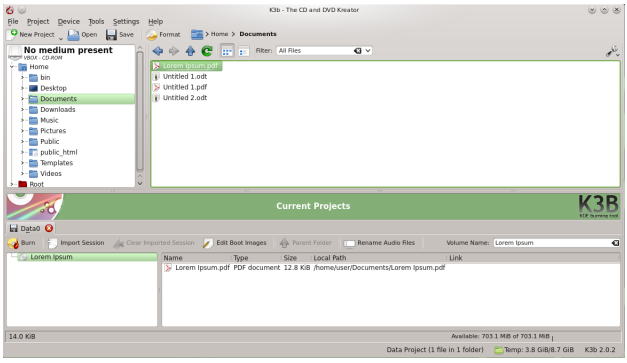
2.16 Creating a CD or DVD

If your computer has a CD or DVD burner, you can burn K3b files on CD or DVD. To create a data CD or DVD, proceed as follows:

1. Press [Alt] + [F2] and enter `k3b`. K3b is opened.

2. In the main menu select *File* > *New Project* > *New Data Project*.

3. Use the tree view in the upper left section of the window to search for the files or folders to be burnt to a disc. As soon as this is displayed in the top right of the window, drag it into the window using *Current projects*.



4 Place a writable CD or DVD into the device and then click on the *Burn icon*.

5 Check the settings on the *Burn tab*. The standard settings should normally work for almost all files. If desired, you can also change further options, such as *Data medium name*, on the *File System* tab.

6 Click on *Burn*.

2.17 Manage your collection of digital images

With digiKam, it is easy to manage digital photos: Download your images from the camera, edit and optimize them, arrange them in albums (or add identifications for simple retrieval, regardless of folders or albums) and archive them on CD or DVD or export them into a gallery with web images.

digiKam also contains a set of useful stacking functions and plug-ins which enable you to convert several images into different formats, rename them or automatically improve the quality of images with the help of functions such as red eye correction, retouching and Hot Pixel removal. Different plug-ins for filters and effects help you to make your digital photos into works of art.

To start digiKam, press [Alt] + [F2] and enter `digikam`. When you start it for the first time, the first time start assistant opens which will guide you through setup. In general, you can confirm all questions with *Next*.

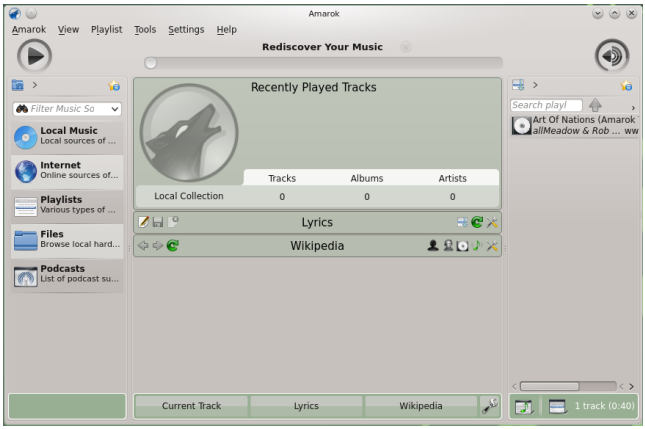


The main window of digiKam contains a tree view on the left hand side and displays thumbnails of the images on the right hand side. To access your images quickly, use the sidebar on the left side which enables you to switch between the different views and to sort the photos according to the *Albums* or *Keywords* for example. To change to the display and editing mode of digiKam, double click on the thumbnail of a photo.

2.18 Managing your music collection

With KDE's Amarok Music Player you can play back different audio formats, create playlists, listen to streaming audio broadcasts on radio stations on the Internet and subscribe to podcasts. The file types supported vary dependent on the Amarok module used.

To start Amarok, press the [Alt] + [F2] key and enter `amarok`.On the first start you have to select the folder which should be searched for your music files. If you want to use the suggested file, simply click on *Yes*.



A sidebar with different views is displayed in the main window of Amarok: your local music collection, some addresses for playing back podcasts or streams from the Internet, your playlists and a file browser. In the middle section, the context browser, information on the current title, the Wikipedia page for interpreters and the song text is displayed as standard. You can see the current playlist in the right hand section of the window. To do so, drag the desired elements from any browser of the sidebar into the area for the playlists. To play back an element, double-click on it in the playlist.

If your music files have the correct identifiers (and contain at least information on the artist and the album), different useful functions of Amarok are available to you. To automatically retrieve images from Amazon for the album cover, select *Tools* > *Cover management* and click on *Fetch missing cover*. The next time you play back a piece of music from this album, the cover is displayed in the context browser and on the display. For further information on the artist, click in the middle section on the *Wikipedia* button Amarok will then start searching for the relevant Wikipedia article.

To configure the use of another module (or other Amarok functions) select *Settings* > *Configure Amarok*.

2.19 Ending Your Session

Log out (so another use can use the computer next), restart the computer or shut it down.

2.19.1 Log out or change user

Click on the *Main Menu* button in your control bar and select *Leave* > *Log Out*. This ends your session but the system remains active. To start a parallel session as a different user, instead select the option *Leave* > *Switch User*.

2.19.2 Restart or shut down the computer

Click on *Leave* and then select one of the following options:

Sleep
: Only available if the computer supports energy management functions. Puts the computer into sleep mode without logging the user off. All data and the session data are stored in the main memory.

Hiberner
: Only available if the computer supports power management functions. Puts the computer into sleep mode without logging the user off. All data and session data are stored on the hard drive before the system goes into hibernation. This protects the data from being lost if a power failure occurs in the meantime.

Redémarrer
: Restarts the computer.

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

2.20 For More Information

You can also find more information on KDE and KDE applications at `http://www.kde.org/` and `http://www.kde-apps.org/`.

openSUSE® provides the tools required by Linux* users for their daily activities. It has a user-friendly graphic environment (the GNOME* Desktop), which communicates with the Linux base system and allows access to files, folders, and programs, as well as their management. openSUSE provides an integrated suite of applications for a wide range of office, multimedia, and Internet activities. It also contains the LibreOffice Suite which enables editing and saving of files in a variety of formats. The office suite is available for several operating systems, meaning you can use the same data across different computing platforms.

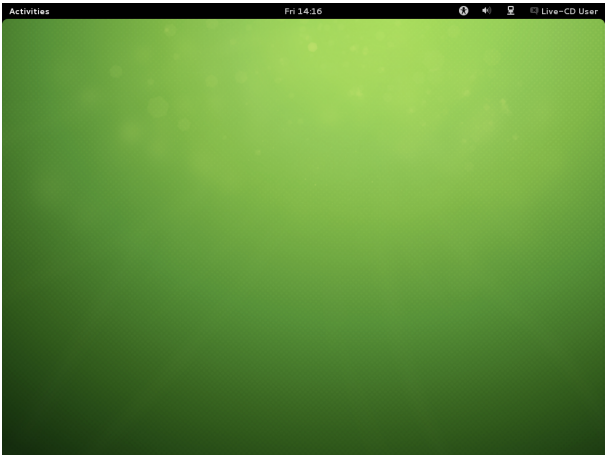
3.1 Getting Started

When starting up your system, you will normally be asked to select your user name and enter your password. If someone else installed your system, ask your system administrator for your user name and password.

The GNOME desktop appears after you first log in. Its appearance depends on whether openSUSE® could activate the 3D functions of your graphics card. Optics and functionality in both cases are the same, only the operation differs slightly.

3.2 The standard GNOME desktop

If openSUSE® could activate the 3D functions, the GNOME desktop notifies you. The largest part is taken up by the initially still empty desktop with the green background image. You will see a narrow black bar on the top edge of the screen. This so-called panel allows you to access all functions and applications. Specifically, it has the following elements:

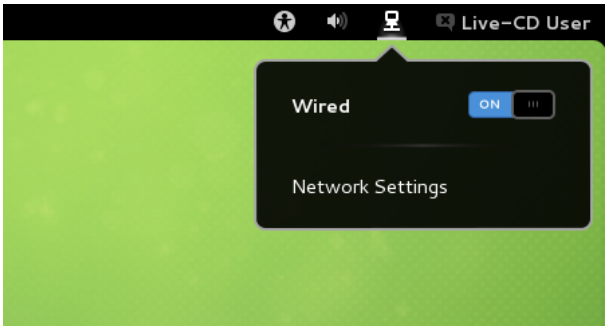


Time, calendar, and schedule planner: The current time is ticking in the middle. When you click on it, a small calendar appears which lists all pending appointments and events on the right hand side. You can call up time and date settings via the relevant buttons or start the calendar application directly (*Open calendar*).

Settings for accessibility: Clicking on the icon with the stylized little man opens different settings for accessibility. You can, for example, increase the contrast, switch on a screen keyboard or enlarge the font. To activate one of the functions, you must only right-click beside the slide switch. An illuminated blue *i* marks a currently active function. Further *settings for accessibility* are behind the similarly named point.

Volume: The icon with the loudspeaker represents media volume. The more semicircles you can see, the louder your loudspeaker becomes. You can adjust the volume by clicking on the icon and dragging the white slider which appears when you press down the mouse button to the left or the right. To silence the sound, push the control all the way to the left. A little cross will be added to the loudspeaker icon. You can find additional settings for audio playback via the *Sound Settings* under the control.

Network: Clicking on the icon directly to the left beside the user name brings up a list with all network connections. The individual connections are sorted here according to their access type. For wireless connections (Wi-fi), small curved lines display the connection quality. A white lock icon indicates that use is only possible after entering a password. You enter it by clicking on the corresponding connection. Whether a connection is active at all and is used by openSUSE® is indicated by the illuminated blue *I* beside to the right. Click on the black switch in order to switch off a connection or explicitly switch it on again. If special settings for your hardware are needed or if problems appear, you can also quickly switch to *Network Settings* on the bottom edge of the list.



User menu: If you click on your name on the top right corner, a menu appears where you manage your user profile and your computer. If you use the immediate news program Empathy, you can change your availability status directly below your name. You can turn off the status messages with the switch underneath. This is useful if, for example, you want to concentrate on your work. You can save your access data for Google and Twitter under *Online Accounts*. You can reach the *System Settings* via the following menu points, where you can, for example, lock the screen for the duration of your lunch break, change to another user account, log off GNOME, or switch off the computer. You can also restart your PC via the last point. If openSUSE® can use the energy-saving functions of your computer, the last menu item is called *Readiness*. This is how you switch your PC to hibernate mode. To go to the *Switch Off* menu item, hold down the [Alt] button.

Activities: Clicking *Activities* on the top left corner of the screen takes you to the so-called activities overview where you start programs and manage those already running. You also reach this view by placing the mouse pointer on the top left corner of the screen or pressing the [Windows] key (also called the [Super] key). You return to the normal view by clicking on *Activities*.

Notifications panel: Whenever you insert a flash drive, finish downloading or some other important thing happens in your system, a corresponding message appears on the bottom edge of the screen. Some of these messages disappear quickly by themselves, others offer equally suitable actions. You can, for example, eject an inserted data CD immediately or open your content in a new window. If you do not wish to do any of the actions, simply click on the title of the message and it will then disappear from the bottom edge. Some messages deliver only a fragmentary short text. To find out more, slide the mouse pointer over the message and it will then expand completely. You can retrieve messages which have been clicked away or which have disappeared too early by clicking on *Activities* in the top left and then hovering the mouse pointer in the lower-right corner of the screen for a few seconds. The so-called notifications panel then appears. Clicking on one of the messages saved there is enough to retrieve the list with the matching actions.

3.2.1 Start Programs

To start an application, first switch to the activities view (click on *Activities* or press the [Windows] button). You now have three options:



About the Dash (Favorites bar): Either click on the program icon in the bar on the left edge of the screen, the so-called Dash (sometimes also inconsistently called a Dock). If you hover over one of the icons with the mouse pointer, GNOME will tell you the name of the pertaining application.

Via the applications view: If the desired program is missing in Dash, click on *Applications*. GNOME will then show all the installed programs. You can move downwards in the list by pulling the ruler which is rather difficult to recognize in the narrow black bar slowly downwards with the left mouse button depressed. If you have installed a lot of programs or the list appears unclear to you, simply click on one of the categories on the right edge. GNOME then filters the diagram according to the corresponding field of application. For example, if you decide on the *Graphics* category, you see only all the available graphics programs.

Via the search box: If you cannot find what you are looking for or when things have to get done quickly, the small search box on the right top will help. As soon as you enter a term there, GNOME will show all the matching applications and system settings. At the same time, you can also consult Wikipedia or Google via the corresponding buttons on the bottom edge. By the way, you do not have to click on the search box first, you can just simply begin typing directly.

3.2.2 Manage Window

After you start an application, GNOME automatically switches back to the workplace. The name of the program and its icon appear in the panel. You can end the application by clicking on it. The *X* on the title bar of its window does the same thing.

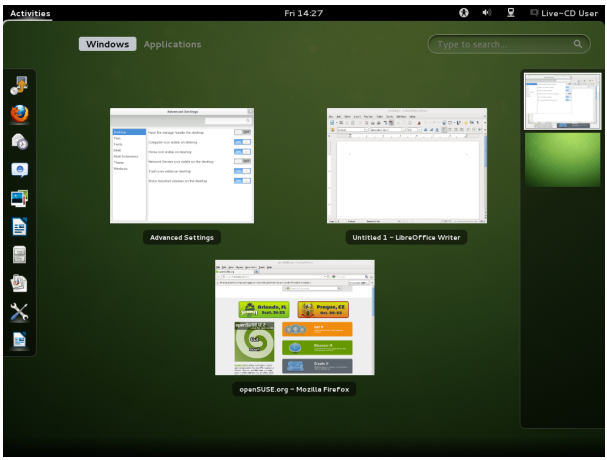
Procédure 3.1 *Maximize Window*

To enlarge a window to the screen size, proceed as follows:

1. Bring the mouse to the title bar of the window (with the program name) and hold and press the left mouse button.
2. Pull the window on the top edge against the panel.
3. As soon as GNOME turns the screen blue, let the mouse button go.

To minimize the window again, follow the same steps but vice versa: Using the mouse, pull away the title bar of the window from the top edge. Alternatively, you can also right-click on the title bar. A menu with the corresponding options then appears. But beware: Differently from other operating systems, *Minimizing* the window changes to *Activities* view.

You can switch quickly between open windows via the [Ctrl] + [Tab]shortcut. *Activities* view gives an overview of all currently open windows. You may need to first click to the *Window* area. Clicking on one of the windows brings it into the foreground, the *X* symbol closes it.



3.2.3 Working with Dash

The small Dash in the *Activities* view not only keeps especially commonly used programs ready, but it can do even more: As soon as you start an application, an icon appears for it in the Dash. A light glow indicates that the application is running and (at least) one window is open.

Right-clicking on an icon opens a small menu which offers different actions depending on the associated program. With *Add to Favorites*, you can place the application icon

permanently in Dash. To remove a program icon from Dash, select *Remove from Favorites*. If you wish to rearrange an icon in the dock, hold the left mouse button pressed above it and then drag it to its new position.

3.2.4 Using Virtual Desktops

GNOME allows you to organize your program window into 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-mails and the calendar and another for word processing and graphics applications.

Procédure 3.2 *Move an Application to Another Virtual Desktop*

To move an open window to another virtual desktop, proceed as follows:



1. Open the *Activities* view.
2. Bring the mouse pointer to the corresponding window and drag it by pressing and holding the left mouse button on the right edge of the screen.
3. A bar with all existing virtual desktops now opens there. Drag and drop the window to the desktop where you would like it to be.
4. Release the left mouse button.

Procédure 3.3 *Switch Virtual Desktops*

You can switch between all existing desktops via [Alt] + [Ctrl] + [Arrow up] and [Alt] + [Ctrl] + [Arrow down]. Alternatively, you can also pull up the activities view:

1. Open the activities view (for example, with the [Windows] key).

2 Bring the mouse to the right edge of the screen. The bar with all existing virtual desktops opens again.

3 Click on the desired virtual desktop on the bar. GNOME now shows all the windows that are there.

4 Click on the virtual desktop once again to finally switch to it.

Procédure 3.4 *Show Window on All Desktops*

You can also show a currently open application on all virtual desktops:

1 Right-click on the title bar of the application.

2 Click on *Always on the Visible Desktop*.

3 You can also very quickly move the window to an entirely specific desktop via the small menu. To do this, select *Move to Another Desktop* and then the corresponding desktop.

3.3 Fallback Mode

If openSUSE® cannot find or activate any 3D graphics card in your computer, GNOME will start in the so-called fallback mode. After *Closing* the corresponding warning instruction, the GNOME desktop then appears with the following elements:



Bottom Panel: The desktop includes a panel at the bottom of the screen. All applications currently open on the desktop appear here. You can access these applications by clicking on their name. Right-clicking on the icon in the bottom-right corner takes you to

the virtual desktop's *Settings*. Provided you have created more than one, a single icon appears for every virtual desktop. In fallback mode, every window has three buttons in its title bar. Through it you can close the window (from right to left), blow up to the screen size or minimize. In the last case, the bottom panel takes up the window.

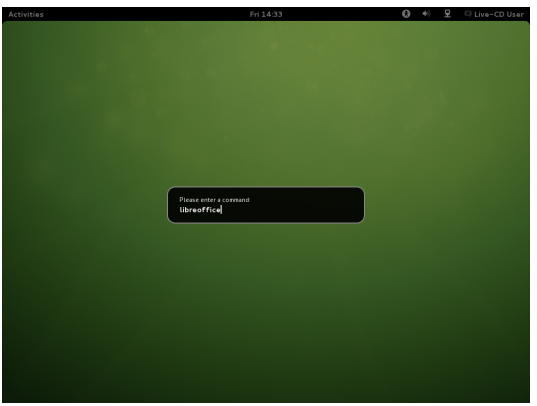
Applications menu: Click on *Applications* on the top left corner to open a menu with all the installed programs. These are classified under different categories for a better overview. Sub-items open automatically as soon as you place the mouse above them.

Places menu: There is also a small menu behind *Places*. Its entries open especially important folders (such as your personal folder or the *Download* folder), show content of connected data carriers and establish a connection to another computer (*connection to server*).

The Panel: The remaining elements on the top black bar function like their counterparts in the normal GNOME mode already described above. Clicking on the time opens a small calendar while the name itself leads to the profile and system settings.

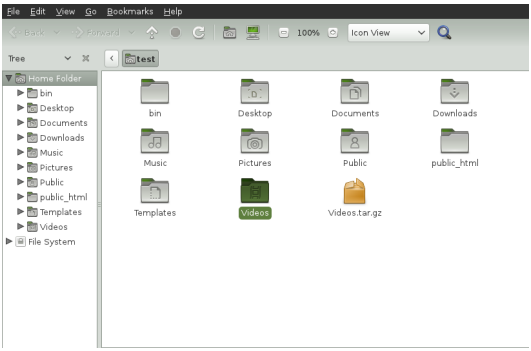
3.4 "Open Application" dialog box“

Press [Alt] + [F2] to open a small dialog box. Enter a command, for example `libreoffice`, and press [Entrée] to start the application. The command to start is often (but not always) the application name written in lower case letters.



3.5 Manage Files, Folders and Media

Use the Nautilus File Manager to create and view folders and documents, run scripts and create CDs of your data. To open Nautilus, change to the Activities view (with the [Windows] key for example) and click on the icon with the filing cabinet in Dash. In fallback mode, instead you open *Places* followed by *Personal Folder*. In any case, a new window appears with the content of your home directory.



The elements of the Nautilus window include the following:

Menu Lets you perform most tasks.

Path bar Lets you find files and folders.

Side bar Lets you navigate or display information about the selected file or folder.

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

3.5.1 Handling Media

What happens exactly if you insert a CD or a DVD into the drive or connect a device (such as a flash drive or a removable disk) depends on the content of the medium. In the simplest case, the file manager opens to display the content of the media.

However, in some cases you can also take a different action. To do that, use the message with a corresponding selection on the bottom edge of the screen. When you insert a DVD, you can for instance decide whether you would like to open it with the media player or the file manager. When in fallback mode, GNOME asks in a new window how to open the data carrier. You can then select a matching application in a drop-down list, *Open Folder* shows the content of the data carrier in a new Nautilus window. Should GNOME not open a window, you can of course start Nautilus manually.

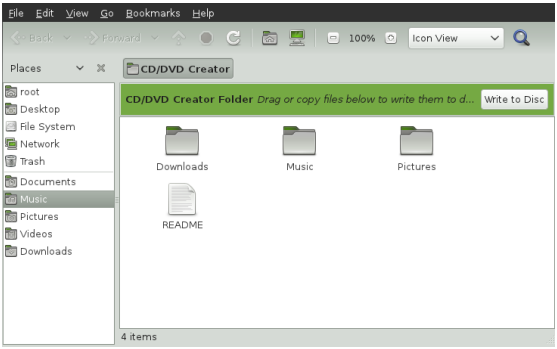
The Nautilus window lists all connected or inserted data carriers in the side pane in the *Devices* area. By clicking on the arrow icon on the right adjacent to its name you can eject the CD or the DVD or the connected flash drive.

3.5.2 Creating a CD or DVD

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

1 Insert an empty data medium and open it in the CD/DVD creator. GNOME then opens a new Nautilus window.

2 Copy the desired files and folders for the CD or DVD into this window.



3 To write data to the CD or DVD, click on *Write to CD/DVD*.

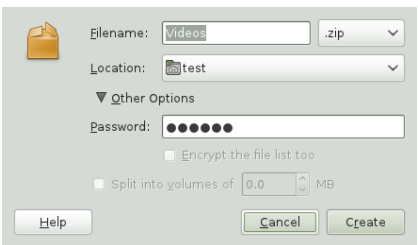
4 Change the corresponding data in the *CD/DVD Creator* window or accept the standard settings. Then click on *Burn*. Now there is no turning back: GNOME will write the files onto the disk. This could take a few minutes, depending on the amount of data being copied and the speed of your burner.

To burn data and audio CDs, you can also use the more extensive burning program Banshee Brasero.

3.5.3 Creating and Unpacking Archives

If you have files you have not used recently but want to keep on your computer, you can compile and compress them in an archive file.

1 To do so, right-click on the folder to be archived in Nautilus and then select *Compress*.



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

3 Select a file name extension from the drop-down list beside to the right. At the same time, you also determine the compression method. Use `.zip` if you would

like to forward the archive to other people or protect it with a password. In Linux the `.tar.gz` format is also widely used.

4 Enter a location for the archive file.

5 If you have decided to use a `.zip` file, you can still click on the plus sign and open *Other Options* and enter a password there. Remember this password well. If you forget it, you will be unable to access the content of the archive.

6 Then click on *Create*.

To unpack an archive created in this way, right-click on the file and select *Extract here*.

3.5.4 Using Bookmarks

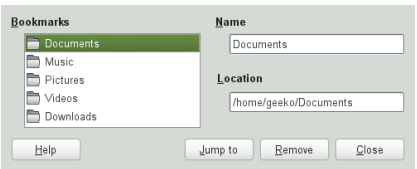
Use bookmarking feature of Nautilus to highlight your favorite folders. A matching entry then appears for quick access in the side pane.

1 Switch (with a double-click) to the folder for which a bookmark should be created.

2 Click on *Bookmark* > *Add bookmark*. The bookmark is added to the list. The folder name is used as the bookmark name.

3 To select an item from the bookmarks list, click on *Bookmark* then click on the desired bookmark in the list.

You can also organize your bookmarks list by clicking on *Bookmark* > *Edit bookmarks* and making your selection in the dialog box.

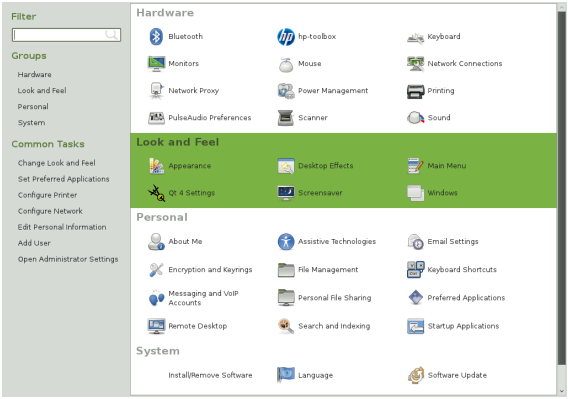


To change the order of your bookmarks, click a bookmark shown in the *Edit bookmarks* dialog box and drag it to the desired location.

3.6 Customizing your Work Environment

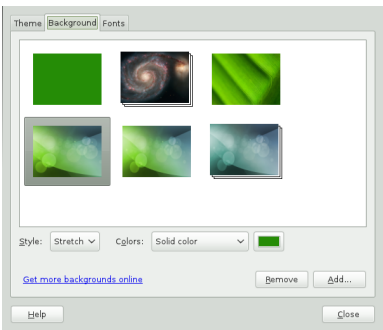
You can find more precise settings to adjust the overall appearance and behavior of your desktop in System Settings. There you can also change keyboard and mouse configurations, regional and language settings, parameters for your Internet and network usage, and more.

To open System Settings, click on your name in the (top) panel and then on *System Settings*.



3.6.1 Changing the Desktop Background

1 Then click on *Background*.



2 If you would like to select a background image from pre-installed images, click on it from the list on the left pane.

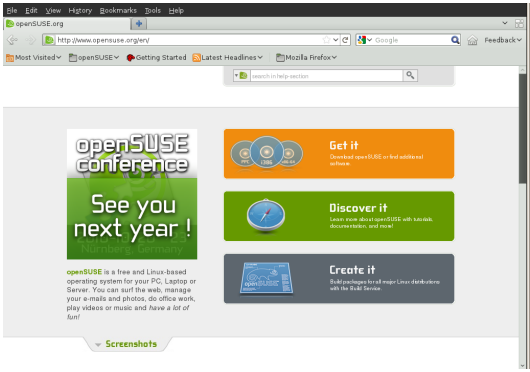
3 To add your own image, click on the plus sign on the bottom left pane corner, search for the desired image and let it *open*.

4 To display a colored background, select from the *Background images* drop-down list the item *Colors and Color Gradients*, highlight the single-color rectangle in the list, click on the bottom right of the window on the button with the color and then choose the desired color.

5 You can return to all system settings via the button with the nine small rectangles on it in the very top left hand corner of the window.

3.7 Browsing the Internet with Firefox

Firefox* is one of the most popular Web browsers. It works like all other browsers, but offers additional functions such as tools for security and data protection.



To start Firefox, switch to the activities view (such as via the [Windows] key) and click on the icon with the globe and the fox in Dash. In fallback mode, you instead open *Applications* > *Internet* > *Firefox*.

Firefox awaits with all modern web technologies, such as use of tabs, pop-up blockers and download and image management. Its easy access to different search engines helps you find the information you need. Enter an Internet address (URL) in the address bar to start browsing.

To open a new empty tab in Firefox click on [Ctrl] + [T] or on the « + » beside to the last tab. Type a new Internet address to browse. To open the page behind a link in a new tab, click on the link with the middle mouse button. Right-clicking on the tab displays a small menu. You can create a new tab, reload the content of a specific tab or all existing tabs, or close a single tab or all tabs. You can also change the order of the tabs by dragging & dropping them to a new position.

3.7.1 Finding Information

1. To start a Web search type the keywords in the integrated search box on the right of the address bar, then press [Entrée]. Firefox will display the results in the active tab.

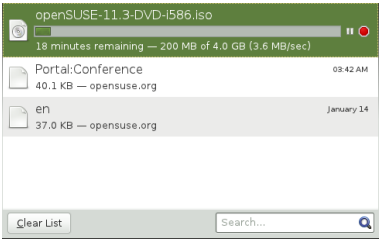
2. If you would like to use a specific search engine, click on the icon with the small down arrow on the search bar.

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

To search for words in the current website, press [Ctrl] + [F]. Enter your keywords in the *search bar* now shown on the bottom of the window. Use the buttons to the right of the search box to switch the search direction or to highlight all results in the text.

3.7.2 Downloading Files from the Web

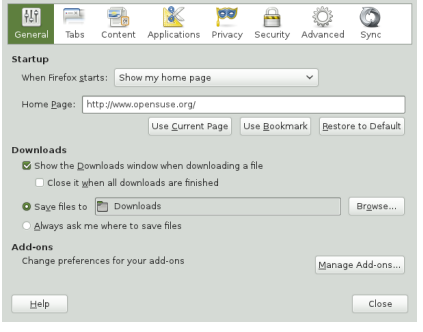
If you download a file with Firefox, the Firefox download manager starts automatically. It then saves the file in the folder previously set in the Firefox *Preferences*. By default, it uses the `Downloads` folder in your home directory. Firefox shows the finished downloads in the *Downloads* window.



Open the downloaded files directly from the *Downloads* window or from the destination folder. Click on *Clean List* to clean up the history of downloaded files.

3.7.3 Configuring Preferences

If you change the standard folder for downloaded files, or activate or want to change the pop-up blocker, click on *Edit* > *Preferences*.



You can configure advanced settings here, such as the appearance, the language, data protection and the tab options. Click on the icons and set the options on each page according to your preferences. To apply the changes click on *Close*.

3.8 Managing Passwords

GNOME allows you to store your passwords and encryption keys in a *Keying*. This is especially useful if you want to store passwords for different websites.

To create a new key ring, proceed as follows:

1. Press [Alt] + [F2] and enter `seahorse`.
2. Select *File* > *New*.
3. Select *Password Key Ring* and continue with *More*.
4. Enter a name for the key ring and click on *Add*. A new window appears.
5. Enter your password, confirm the password, and click *Continue*.

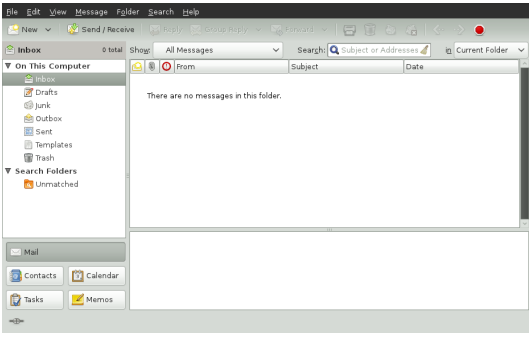
You can create as many key rings as you like. For example, you can create one for private use and a second one for business. To store passwords in your key ring, proceed as follows:

1. Press [Alt] + [F2] and enter `seahorse`.
2. Select *File* > *New*.
3. Select *Save password* and continue with *More*.
4. Select the key ring, enter a description and the password.
5. Finally click on *Add*.

3.9 E-mail and Calendar

To read and manage your mails, contacts, appointments, and tasks, openSUSEcontains the program Evolution™, which allows you to conveniently save, organize, and call up your personal information.

With its extensive support for communications and data interchange standards, Evolution can work with existing corporate networks and applications, including Exchange*.



To start Evolution, switch to the activities view (for example, via the [Windows] key) and click on the letter icon. In fallback mode, select *Applications* > *Internet* > *Evolution E-Mail and Calendar*.

If you are starting Evolution for the first time, you will be asked to answer a series of questions. The queried information is needed to set up your e-mail account and helps you import e-mail messages from your old e-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 e-mail tools are available on the toolbar on the left side of the program window.

Press [Ctrl] + [N] to open a new item in the area of Evolution where you are working. This creates a new message in e-mail. In the address book [Ctrl] + [N] creates a new contact card and a new appointment in the calendar.

If you would like additional information, click in any window on *Help* > *Content*.

3.10 Instant Messaging with Empathy

Empathy is a multi-protocol 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 Empathy, you can log in to multiple accounts on multiple IM networks simultaneously. Empathy also supports many features of various networks such as video chat, file transfer, away messages and typing notification.

To start Empathy, switch to the activities view (for example, via the [Windows] key) and click on the icon with the speech balloon and the blue smiley in the Dash. In fallback mode, try *Applications* > *Internet* > *Empathy Instant Messenger Service*.

To add an account to Empathy, use the configuration assistant, then select *Edit* > *Accounts*. Click on the plus symbol, select the desired protocol and enter your registration information into the relevant fields. Click on *Apply* and *Close*, in order to activate your IM account.

Add contacts by clicking on *Chat* > *Add Contacts*. Select the account, the *Identifier* (ID), and the *alias* information that is relevant to the new content, and confirm with *Add*. 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 on the desired contact and type your message in the lower part of the chat window. Press [Entrée] to send the message. The upper part of the window displays the messages you have sent and received.

3.11 Starting LibreOffice

LibreOffice offers comprehensive and powerful office programs, including a text processing program, a table calculation application, a program for the creation of

presentations, a vector graphic program and a database application. Because LibreOffice is available for a number of operating systems, you can use the documents across different operating systems.

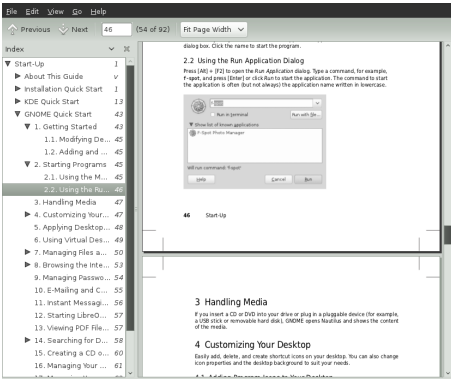
To start LibreOffice switch to the activities view (for example, by pressing the [Windows] key) and then click on the blue document. Alternatively, type `writer` in the search box and press [Entrée]. In fallback mode, click on *Applications* > *Office* > *LibreOffice Writer*. If you would like to create a new document, click on *File* > *New* and select the document type. To open an existing document, select *Open*, then select the file you want from the file system.

3.12 Viewing PDF Files and Other Documents

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

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

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



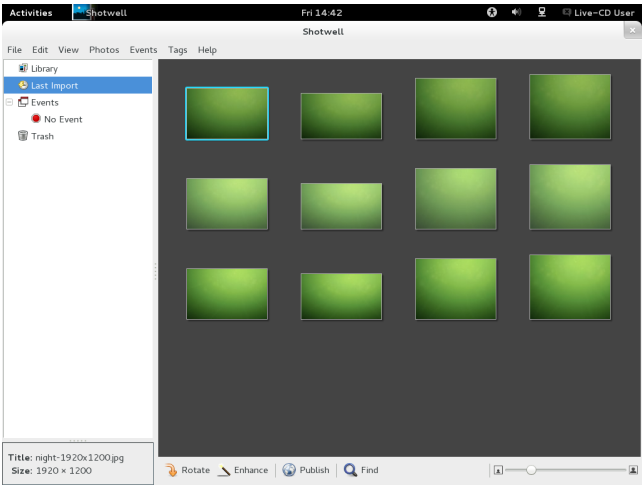
2 Use the navigation icons at the top of the window to navigate through the document. If the PDF document contains bookmarks, you can access them to the left of the side pane.

3.13 Managing Your Digital Image Collection

Shotwell is a management tool for digital image collections. You can assign various category tags to your images. It also offers many practical image editing options. For

example, you can remove red eye, crop an image to size as well as adjust brightness and colors. Shotwell supports all important file formats, such as JPEG, PNG, TIFF and various manufacturer-specific RAW formats, for example. With Shotwell, you can also export photos to your Flickr, Picasa or Facebook account.

To access Shotwell, open the activities view (for example, by pressing the [Windows] key) and then click on the icon with the images in Dash. In fallback mode, you can find Shotwell via *Applications* > *Graphics* > *Shotwell Photo Manager*. After you pass through the welcome window by clicking *OK*, Shotwell immediately runs through your image folder and automatically add all the images stored in it to its library.



Shotwell shows thumbnails of the images to the right in the window. As soon as you click one of them, you can *rotate it*, re-edit it (*enhance it*) or *publish it* in an Internet service with the buttons on the lower edge. The side bar to the left offers different views of your photo portfolio. Provided the photos were stamped with a date, you can limit the display to photos from a specific year via *Events* for example.

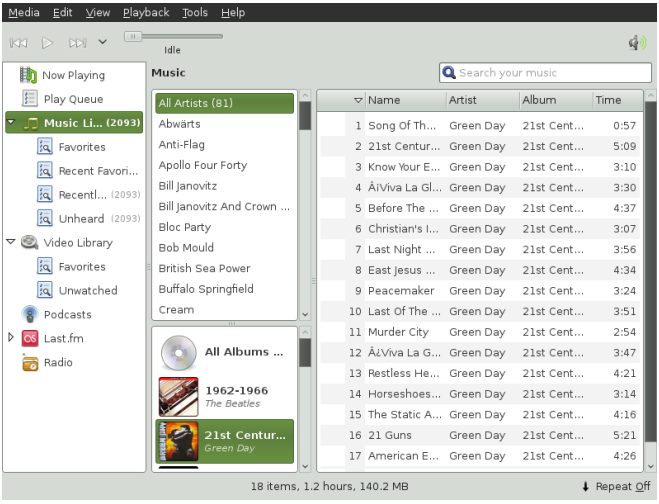
You import new photos via *File* > *Import from Folder*. Shotwell always copies them to your image folder. If you would like to set another folder for this, go to *Edit* > *Preferences* and choose the corresponding folder under *Import Photos to*.

3.14 Managing Your Music and Videos

Use Banshee to import CDs, synchronize 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 switch to activities view (for example, via the *Windows* key) and click on the icon with the b in the form of a note. In fallback mode click on *Applications* > *Multimedia* > *Banshee Media Player*.

When you start the Banshee for the first time, you need to import your media to make them available in the library. From the menu, call up *Media* > *Import Media*. Select an import source and click on *Choose Files* or *Choose Folders*. 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 has an integrated CD player. When you insert a music CD, your CD title appears in the left panel. Select the title and click on the *Play button* to play your full CD.

To create Audio and MP3 CDs, select the desired song and then click on *Edit* > *Write CD* in the menu .

3.15 Ending Your Session

Once you have finished your work on the computer, you can log out. However, you can also restart the system or shut down the computer.

3.15.1 Logging Out or Switching Users

Click on your user name on the top right in the panel and 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 Likewise returns you to the login screen but, in contrast to *Log Out*, the applications started continue running in the background. This feature is subsequently offered if another person has to log in to your computer for a short while using his or her own user account.

Lock Screen Turns the screen black. You must enter your password to close it. This prevents unauthorized persons from working on or playing around with your computer during your absence.

3.15.2 Restarting or Shutting Down the Computer

If openSUSE® can use the power-saving functions of your computer, your PC can switch to sleep mode via the *Suspend* item. Otherwise, GNOME offers you *Power Off.... Power Off...* can also be accessed by holding down the [Alt] button while the menu is open. If click on it, a new window offers you the following actions:

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

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

Hibernate This feature puts your computer into power-saving mode. The PC consumes only a small amount of electricity while in this mode. After you switch the PC on again, your work environment as you had left it becomes available again. In particular, all documents remain open.

3.16 For More Information

Additional information on GNOME and GNOME applications can be found under http://www.gnome.org/.

LibreOffice is a powerful Office suite, consisting of word processing, spreadsheet, presentation, database and drawing programs.

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).

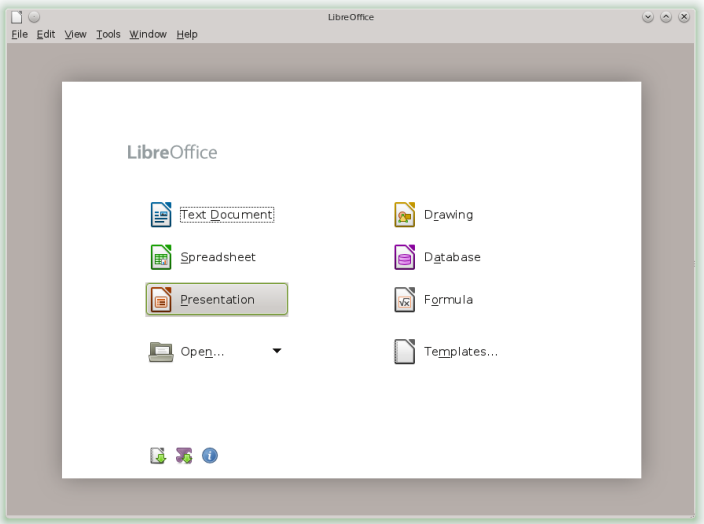
4.2 LibreOffice Modules

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.

Tableau 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 [Alt] + [F2], type `libreoffice` and press [Enter]. In the appearing window, 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 file, select *File* > *Save* or *Save as* from the *Menu Bar* or use the shortcut [Ctrl] + [Maj] + [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:

Tableau 4.2 *Navigation Shortcuts*

Action	Shortcut
Delete to the End of the Word	[Ctrl] + [Suppr]
Delete to the Beginning of the Word	[Ctrl] + [<—]
End of Document	[Ctrl] + [End]
End of Line	[End]
Select All	[Ctrl] + [A]
Start of Line	[Deébut]
Top of Document	[Ctrl] + [Deébut]
Word Left	[Ctrl] + [←]
Word Right	[Ctrl] + [→]

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* and the wizard you wish to use. Then 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`). Then choose a Template and click on *Open*. Now 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 *File* > *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 *Save*.

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.



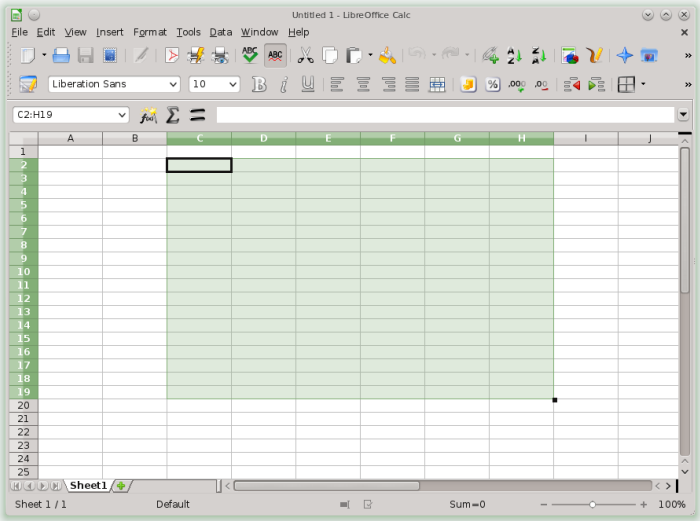
To enter data, select a cell with your mouse pointer and begin typing. To move around cells, use the arrow keys. [→|] moves to the next cell, skipping over the contents of the cell and [Maj] + [→|] moves backwards.

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.

Procédure 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*.

Procédure 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.

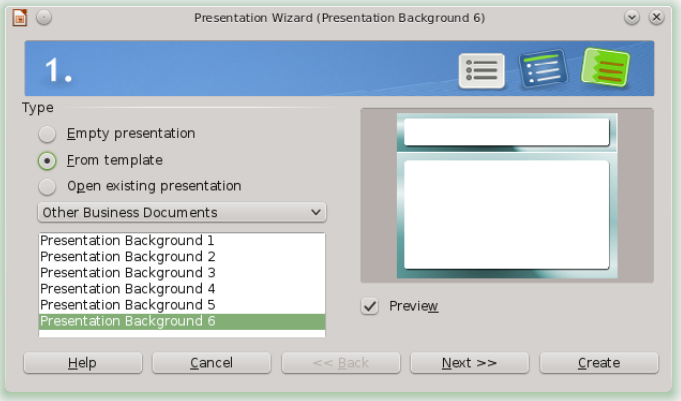
4.6.1 Using Presentations with Impress

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 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:

Procédure 4.3 *Creating a Database*

1 Click *File* > *New* > *Database*.

2 Select Create a new database and proceed with *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*.

Procédure 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.

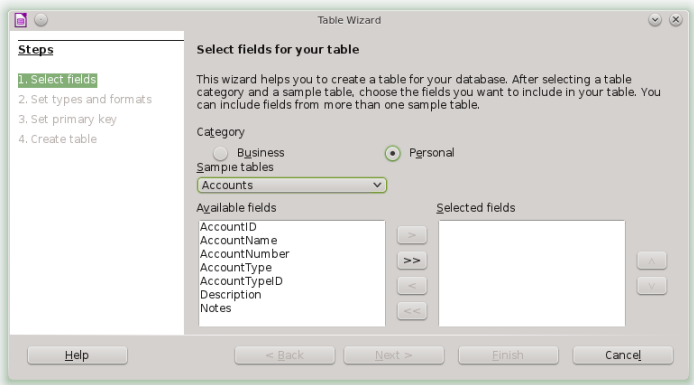
4 Click the single arrow icon to move the selected items to the *Selected fields* menu.

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.

6 Click on *Finish* to create a table for your address book, which you can now insert actual data into.



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.

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.

Procédure 4.5 *Creating a Graphic*

1. Click [Alt] + [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.

Procédure 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 3.6 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*.

REMARQUE

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

Partie II. Managing and Updating Software

Utilisez l'outil de gestion de logiciels YaST pour chercher pour les composants logiciel que vous souhaitez ajouter ou enlever. YaST résoud toutes les dépendances pour vous. Pour installer des paquets non livrés avec le média d'installation, ajoutez des dépôts de logiciels supplémentaires pour votre configuration et laissez YaST les gérés. Gardez votre système à jour avec l'application de mise à jour.

Changez la collection de logiciels de votre système avec le module de gestion des logiciels de YaST. Ce module YaST est disponible en trois versions de boîte à outils : Qt(pour les environnements KDE), GTK+ (pour les environnements GNOME), et ncurses (fournissant une interface utilisateur pseudo-graphique en mode texte). Ce chapitre décrit les versions Qt et GTK+ pour des détails sur le ncurses de YaST, voir le Chapitre 12, *YaST in Text Mode* (p. 143).

ASTUCE : Changement de la version de la boîte à outils.

Par défaut, YaST est démarré avec la boîte à outils correspondant à votre environnement (GTK+ sous GNOME, Qt sous KDE). Pour modifier ces réglages par défaut, changez la variable `WANTED_GUI` dans `/etc/sysconfig/yast2` pour `qt` ou `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` parameter, respectively. For example: `yast2 --gtk`.

REMARQUE : Confirmation et vérification des modifications.

Pendant l'installation, la mise à jour ou l'enlèvement d'un paquet, tout changement dans le gestionnaire de logiciel n'est pas appliquée immédiatement mais seulement après les avoir confirmer avec *Accepter* ou *Appliquer* respectivement. YaST maintient une liste de toutes les actions, vous permettant de les vérifier et les modifier avant de les appliquer sur votre système.

5.1 Définitions

Dépôt
: Un répertoire local ou distant contenant des paquets, et des informations supplémentaires sur ceux-ci (métadonnées)

Alias (de dépôt)
: Le nom court d'un dépôt utilisé par plusieurs commandes zypper. Les alias peuvent être choisis par l'utilisateur au moment de l'ajout d'un dépôt et doit être unique.

Produit
: Représente un produit complet, par exemple openSUSE®.

Modèle
: Un modèle est un groupe de paquets installables dédié à certain usage. Par exemple, le modèle `Laptop` contient tous les paquets nécessaires aux environnements informatiques itinérants. Les modèles définissent des dépendances de paquets (comme des paquets requis ou recommandés) et viennent avec une présélection de sélection de paquets marqués pour l'installation.Ceci assure que les plus importants paquets nécessaires pour un certain usage sont disponibles sur votre système après l'installation du modèle. Cependant, tous les pacquets ne sont pas nécessairement préselectionnés dans un modèle pour l'installation et vous pouvez sélectionner ou désélectionnerles paquets dans un modèle selon vos besoins et vos souhaits.

Paquet
: Un paquet est un fichier compréssé au format `rpm` qui contient les fichiers pour un programme particulier.

Patch
: Un correctif comprend un ou plusieurs paquets et peut être appliqué au moyen de deltarpms. Il peut être aussi introduire des dépendances pourdes paquets qui ne sont pas encore installés.

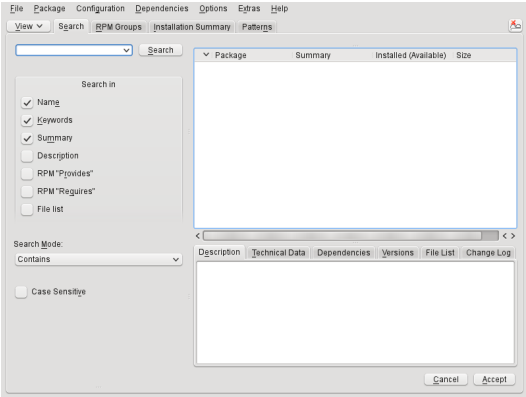
Solution
: Terme générique pour les produits, modèles, paquets ou correctifs. Les types les plus utilisés de resolvable sont les paquets et les correctifs.

rpm-delta
: Un rpm-delta se compose seulement de la différence entre les deux versions d'un paquet, elle a donc une taille de téléchargement minimale. Avant d'être installée, le paquet RPM entier est reconstruit sur la machine locale.

Dépendances d'un paquet
: Certains paquet sont dépendant d'autres, tel que des librairies dynamiques. En d'autres termes, un paquet peut `nécessiter` d'autres paquets—si le paquet requis n'est pas disponible, le paquet ne peut pas être installé. En plus des dépendances (exigées pour le paquet) qui doivent être complètement remplies, quelques paquets `recommandent` d'autres paquets. Ces paquets recommandés ne sont seulement installés que si ils sont réellement disponibles, sinon ils sont juste ignorés et le paquet les recommandant est néanmoins installé.

5.2 Utilisation de l'interface KDE (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 Vues pour la recherche de paquets ou de modèles.

Le gestionnaire de logiciels YaST peut installer des paquets ou des modèles depuis tout dépôt courant disponible. Il offre différentes vues et filtres pour faciliter la recherche de logiciels. La vue *Rechercher* est la vue par défaut de la fenêtre. Pour changer de vue, cliquez *Vues* et sélectionnez une des entrées du menu déroulant. La vue sélectionnée s'ouvrira dans un nouvel onglet.

Modèles
: Lists tous les modèles disponibles pour l'installation sur votre système.

Groupes de paquets
: Liste tous les paquets triés par groupes tels que *Graphiques*, *Programmation*, or *Securité*.

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

Langues
: Filtre tous les paquets nécessaire pour ajouter une nouvelle langue pour le système.

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

Chercher
: Lets you search for a package according to certain criteria. Enter a search term and press [Entrée]. 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.

Résumé de l'installation
: 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 [Maj] + [F1] for details on the status flags.

ASTUCE : 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 Installation et retrait de paquets ou de modèles.

Certains paquets sont dépendants d'autres paquets, tel que des bibliothèques partagées. D'un autre côté, quelques paquets ne peuvent co-exister avec d'autres sur un systèmes. Si possible, YaST résoud automatiquement les dépendances et les conflits. Si votre choix produit des conflits de dépendance qui ne peuvent être résolus, vous aurez besoin de les résoudre manuellement de la façon décrite dans la Section 5.2.4, « Vérifier les dépendances logiciellles » (p. 81).

REMARQUE : Suppression de paquets

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. Cherchez les paquets de la manières décrites à la Section 5.2.1, « Vues pour la recherche de paquets ou de modèles. » (p. 77).

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 [Maj] + [F1] for help.

   ASTUCE : Appliquez une action à tous les paquets listés

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

   ---

3. pour installer un modèle, clic-droit sur le nom de modèle et choisir *Installer*.

4. Il n'est pas possible d'enlever un modèle en soi. A la place, sélectionner les paquets d'un modèle que vous voulez supprimer et marquez les pour la suppression.

5 Afin de sélectionner plusieurs paquets, répétez les étapes mentionnées ci-dessus.

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 Afin de rétabir le statut d'un paquet, faites un clic-droit sur le paquet et sélectionnez une des entrées suivantes : *Garder* si le paquet a été programmé pour être effacé ou mis à jour, ou *Ne pas installer* si celui-ci a été programmé pour l'installation. Pour abandonner tous les changements et fermer le gestionnaire de logiciels, cliquez *annuler* et *Abandon*

8 Quand vous avez fini, cliquez sur *Accepter* pour appliquer vos changements.

9 Dans le cas où YaST trouve des dépendances sur d'autres paquets, une liste des paquets qui on été choisis en plus pour l'installation, la mise à jour ou la suppression est présentée. Cliquez *Continuer* pour les accepter.

Après que tous les paquets ont été installés, mis à jour ou supprimé, le gestionnaire de logiciels YaST termine automatiquement.

REMARQUE : Installation des paquets sources.

Intaller des paquets sources avec le gestionnaire de logiciels YaST n'est pas possible pour le moment. Utilises l'outil en ligne de commande `zypper` pour cela. Pour plus d'informations, voir Section 9.1.2.1, « Installing Source Packages » (p. 107).

5.2.3 Mise à jour de paquets

Au lieu de mettre à jour les paquets individuellement, vous pouvez aussi faire les mises à jour de tous les paquets installés de tous les paquets d'un dépôt particulier. Quand vous faites ce type de mise à jour de masse, les aspects suivants sont généralement pris en compte :

- 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.

Lequel de ces aspects a la plus haute importance pour le choix des paquets candidats à la mise à jour en regard de l'option de mise à jour que vous avez choisi.

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

Tous les dépôts sont vérifier pour déterminer s'ils sont candidats pour une mise à jour, en utilisant la politique suivante : YaST premièrement essaie de

restreindre la recherche aux paquets de même architecture et du même fournisseur que celui déjà installé. Si la recherche est positive , le « meilleur » candidat à la mise à jour parmi eux est sélectionné selon le processus ci-dessus. Cependant , si aucun paquet comparable du même fournisseur ne peut être trouvé., la recherche est étendue à tous les paquets avec la même architecture. Si encore une fois aucun paquet comparable n'est trouvé, tous les paquets sont pris en compte et le « meilleur » candidat est sélectionné selon les critères suivant :

1. Priorité au dépôt : Préfère le paquet depuis le dépôt ayant la plus haute priorité.

2. Si plus d'un paquet resultent de cette sélection, choisissez celle avec le « meilleure » architecture (meilleur choix : celui correspondant à l'architecture installée; sinon: x86_64 > i686 > i586).

Si le paquet résultant a un numéro de version que celui installé, le paquet installé sera mis à jour et remplacer par le candidat sélectionné.

Cette option essaie d'éviter le changement d'architecture ou de fournisseur pour les paquet installés, mais en certaines circonstances, ces changements sont tolérés.

REMARQUE : Mise à jour inconditionnelle

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

---

2 Pour vous assurez que les paquets pour une mise à jour de masse proviennent d'un dépôt spécifique:

2a Choisir le dépôt depuis lequel faire la mise à jour comme décrit à la Section 5.2.1, « Vues pour la recherche de paquets ou de modèles. » (p. 77) .

2b Sur la droite de la fenêtre, cliquez *Synchroniser les paquets systèmes aux versions de ce dépôt*. Ceci permet explicitement à YaST de changer le fournisseur du paquet au moment du remplacement des paquet.

Dès que vous avez terminé avec *Accepter*, tous les paquets installés seront remplacé par les paquets issuent de ce dépôt, si ils sont disponibles. Cela peut conduire à des changements de l'architecture, du fournisseur et même du déclassement de certains paquets.

2c Pour s'abstenir de cela, cliquez *Annuler la synchronisation des paquets systèmes au version de ce dépôt*. Notez que l'annulation

aura lieu seulement au moment où vous presssez le bouton *Accepter*.

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 Si toutes les options sont mise en conformité à vos souhaits, confirmez vos changements avec *Accepter* pour commencer la mise à jour de masse.

5.2.4 Vérifier les dépendances logiciellles

Beaucoup de paquets dépendent d'autres. Si un paquet, par exemple, utilise une librairie partagée, il est dépendant du paquet fournissant cette librairie. D'autre part quelques paquets ne peuvent coexister avec d'autres, ceci causant des conflits (par exemple, vous ne pouvez seulement installé qu'unseul agent de transfert de mail : sendmail ou postfix). Au moment de l'installation ou de la supression des logiciels, le gestionnaire de logiciels assure qu'aucune dépendance ou conflit ne soit irrésolu afin d'assurez l'intégrité du système.

Au cas où il existe seulement une solution pour résoudre une dépendance ou un conflit, il est résolu automatiquement. Des solutions multiples produisent des conflits qui ne peuvent être résolus que manuellement. Si la résolution d'un conflit implique un changement de fournisseur ou un changement d'architecture, elle a besoin d'êtres faite aussi manuellement.Quand vous cliquez sur *Accepter* pour appliquer tous les changements dans le gestionnaire de logiciels, vous obtenez un vue d'ensemble de toutes les actions entreprise par le résolveur automatique qui ont eu besoin d'une confirmation.

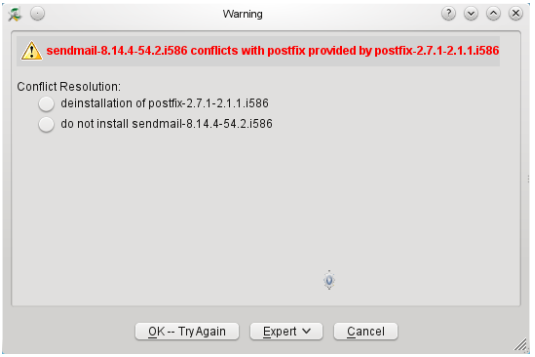
By default, dependencies are automatically checked. A check is performed every time you change a package status (for example, by marking a package for installation or removal). This is generally useful, but can become exhausting when manually resolving a dependency conflict. To disable this function,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

Pour revoir une dépendance de paquets, faites un clic droit dessus et choisir *Montrer les informations du solveur*. Une carte montrant les dépendances s'ouvre. Les paquets qui ont été déjà intallés sont affichés dans une fenêtre verte.

REMARQUE : Résolution manuelle des conflits

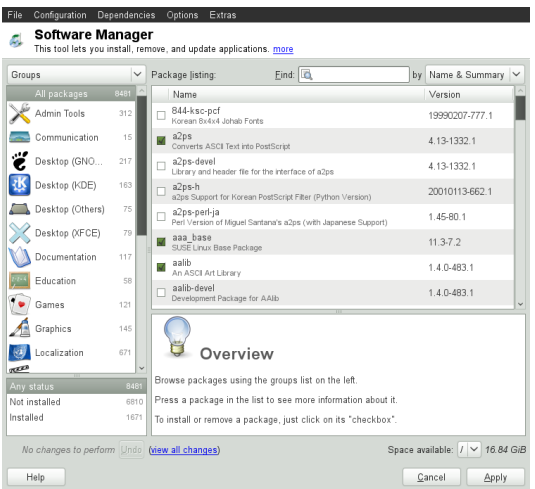
Si vous n'êtes pas très expérimenté, suivez les recommandations de YaST produitent lors de la manipulation de paquets conflictuels, car vous ne pouriez pas pouvoir les résoudre. Gardez à l'esprit que tous les changements que vous faites peuvent potentiellement déclencher d'autres conflits, ainsi vous pouvez facilement vous retrouver avec un nombre sans cesse croissant de conflits. Au cas où cela vous arrive, *Annuler* Le gestionnaire de logiciel, *Abandonner* tous vos changements et recommencez.

Figure 5.1 *Gestionnaire de conflit du gestionnaire de logiciels*



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 [Entrée]. 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 [Ctrl] 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, « Vérifier les dépendances logicielles » (p. 81).

1. Search for packages as described in Section 5.3.1, « Views for Searching Packages or Patterns » (p. 82).

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.

ASTUCE : 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.

REMARQUE : 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 » (p. 107).

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 Chapitre 6, *YaST Online Update* (p. 95).

   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 » (p. 82) .

   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 resolved

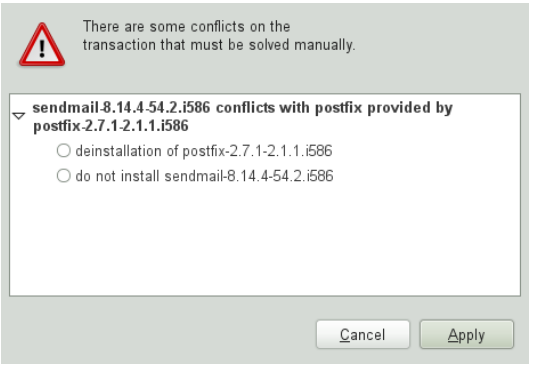
manually. If solving a conflict involves a vendor or architecture change, it also needs to be solved manually. When clicking *Apply* to apply any changes in the Software Manager, you get an overview of all actions triggered by the automatic resolver which you need to confirm.

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* > *Auto-check*. Manually perform a dependency check with *Dependencies* > *Check Now*. A consistency check is always performed when you confirm your selection with *Apply*.

REMARQUE : 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 manager. 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 12.3 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.

AVERTISSEMENT : 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 » (p. 88) 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, « Utilisation de l'interface KDE (Qt) » (p. 76) or in Section 5.3, « Using the GNOME Interface (GTK+) » (p. 82).

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).

Procédure 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 Down-*

loaded 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 gearwheel with a green arrow. To start Software Updater manually, choose *System Settings* > *Software Management* > *Software Updates* from the main menu. Alternatively, press [Alt] + [F2] and enter `kpk_update`.

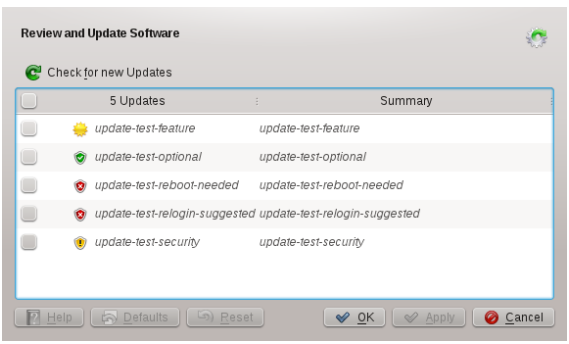
REMARQUE : Icon Visibility

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 Chapitre 6, *YaST Online Update* (p. 95) 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 [Alt] + [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 12.3* 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*.

REMARQUE : Icon visibility

The applet icon is only visible if the following conditions are met:

- patches are available
- the GUI was not started as user `root`
- the GUI was not started in a VNC session

To start the update viewer even if no applet icon is visible, press [Alt] + [F2] and enter `gpk-update-viewer`.

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.

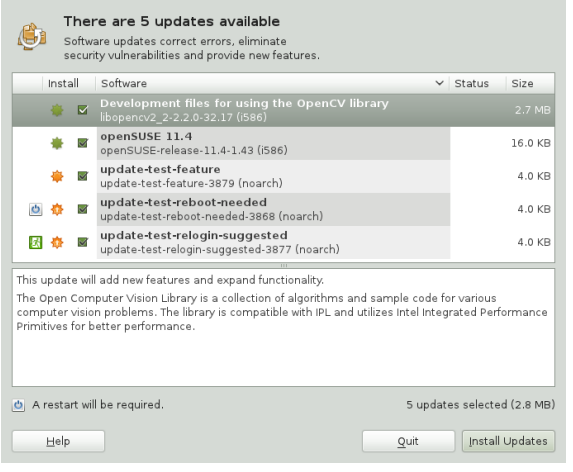
5.5.2.1 Installing Patches

Procédure 5.2 *Installing Patches*

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

1. 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 Chapitre 6, *YaST Online Update* (p. 95) 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 » (p. 89) 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 » (p. 86).

openSUSE provides updates with different relevance levels:

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

Recommended Updates
: Fix issues that could compromise your computer.

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

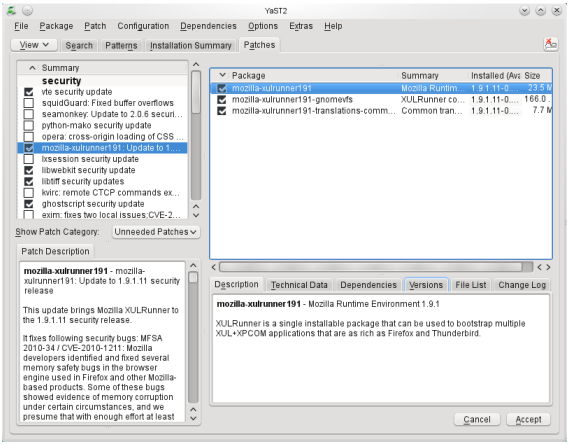
6.1 The Online Update Dialog

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 [Maj] + [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.

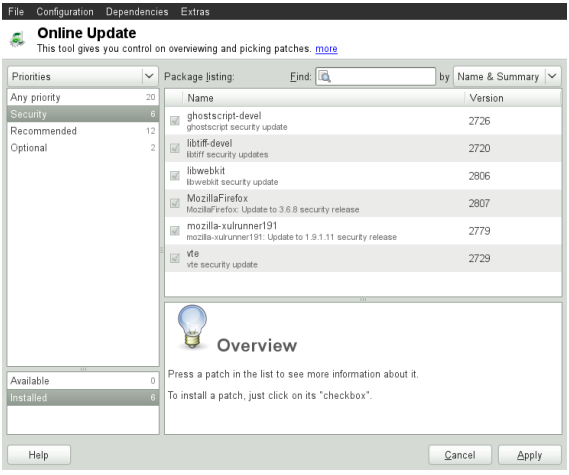
Select an entry in the *Summary* section to view a short *Patch Description* at the bottom left corner of the dialog. The upper right section lists the packages included in the

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 once or to manually select the patches that you want to apply to your system. You may also revert patches that have been applied to the system.

By default, all new patches (except `optional` ones) that are currently available for your system are already marked for installation. They will be applied automatically once you click *Accept* or *Apply*.

Procédure 6.1 *Applying Patches with YaST Online Update*

1. Start YaST and select *Software* > *Online Update*.
2. To automatically apply all new patches (except `optional` ones) that are currently available for your system, proceed with *Apply* or *Accept* to start the installation of the preselected patches.
3. 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) » (p. 95) and Section 6.1.2, « GNOME Interface (GTK) » (p. 97).
   - 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

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

ASTUCE : Disabling deltarpms

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

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

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.

Procédure 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*.

7 Installing Packages From the Internet

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 Build Service hosts several hundred ones and a lot of third party repositories exist, too—see . 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 Package Search lets you query almost all known repositories for . You can directly install any package found by the package search module. Trusting External Software Sources Before installing from any external software repository, make sure it can be trusted. is not responsible for any potential problems arising from software installed from third-party software repositories.

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 » (p. 88) 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 Chapitre 5, *Installation et Désinstallation de Logiciels* (p. 75) 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, « Définitions » (p. 75).

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 URL_TO_FIREFOX_RPM
```

Replace *URL_TO_FIREFOX_RPM* with the actual download URL such as `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.

AVERTISSEMENT : 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 » (p. 111) 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 » (p. 171).

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 » (p. 111). 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 » (p. 173).

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.

REMARQUE : 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:

Exemple 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 Exemple 9.1, « Zypper—List of Known Repositories » (p. 111), 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 complete and detailed command reference, including `cheat sheets` with the most important commands, and information on how to use Zypper in scripts and applications, refer to `http://en.opensuse.org/SDB:Zypper_usage`. A list of software changes for the latest 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`.

ASTUCE : 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 Delta RPM Packages

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

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

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

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

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

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

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

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

9.2.4 RPM Queries

With the `-q` option `rpm` initiates queries, making it possible to inspect an RPM archive (by adding the option `-p`) and also to query the RPM database of installed packages. Several switches are available to specify the type of information required. See Tableau 9.1, « The Most Important RPM Query Options » (p. 116).

Tableau 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 Exemple 9.2, « rpm -q -i wget » (p. 117).

Exemple 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 Exemple 9.3, « Script to Search for Packages » (p. 118). Pass the partial filename to the script shown as a parameter when running it.

Exemple 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:

Tableau 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`.

9.2.5 Installing and Compiling Source Packages

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

REMARQUE : 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`.

AVERTISSEMENT

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 -b`*`X`* `/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.

9.2.6 Compiling RPM Packages with build

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

```
cd /usr/src/packages/SOURCES/
mv ../SPECS/wget.spec .
```

```
build --rpms /media/dvd/suse/ wget.spec
```

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

The `build` script offers 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.

9.2.7 Tools for RPM Archives and the RPM Database

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

A full-featured package manager is available as a YaST module. For details, see Chapitre 5, *Installation et Désinstallation de Logiciels* (p. 75).

Partie III. Administration

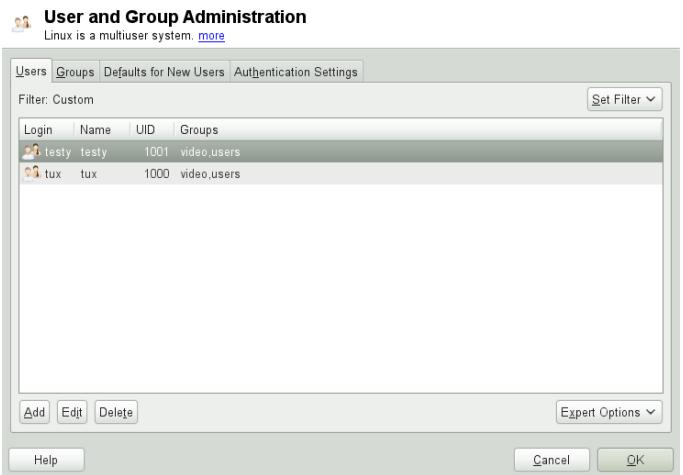
During installation, you chose a method for user authentication. This method is either local (via `/etc/passwd`) or, if a network connection is established, via NIS, LDAP, Kerberos or Samba (see section « Créez un nouvel utilisateur » (Chapitre 1, *Installation avec YaST*, ↑Reference)). You can create 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 » (p. 126). 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 » (p. 128).

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 » (p. 133).

Assigning Users to Groups
: Learn how to change the group assignment for individual users in Section 10.5, « Assigning Users to Groups » (p. 133).

Managing Groups
: From the *Groups* tab, you can add, modify or delete existing groups. Refer to Section 10.6, « Managing Groups » (p. 134) 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 » (p. 135).

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.

REMARQUE : 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 » (p. 128).

Procédure 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 » (p. 128).
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.

ASTUCE : 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.

Procédure 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.

AVERTISSEMENT : 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:

Procédure 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.

REMARQUE : 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.

AVERTISSEMENT : 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 » (Chapitre 10, *Encrypting Partitions and Files*, ↑Security Guide).

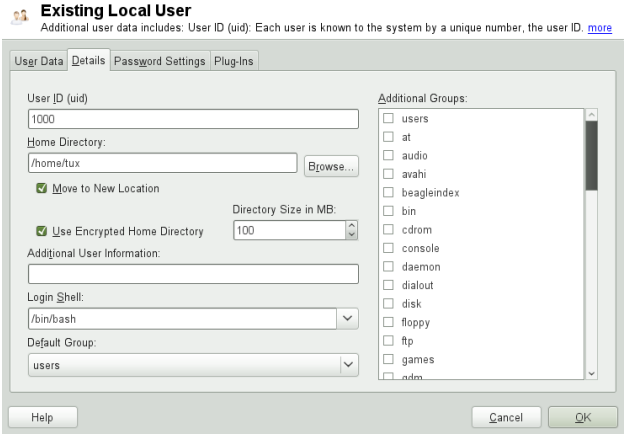
Procédure 10.4 *Creating Encrypted Home Directories*

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.

Procédure 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 Chapitre 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.

Procédure 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*.

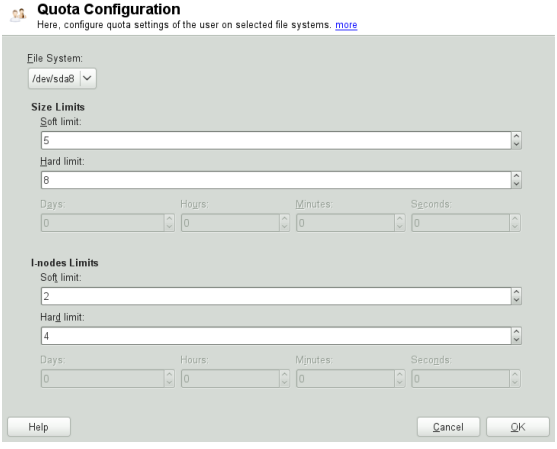
Procédure 10.7 *Setting Up Quotas for Users or Groups*

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 Chapitre 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 » (p. 133).

Procédure 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.

Procédure 10.9 *Creating and Modifying Groups*

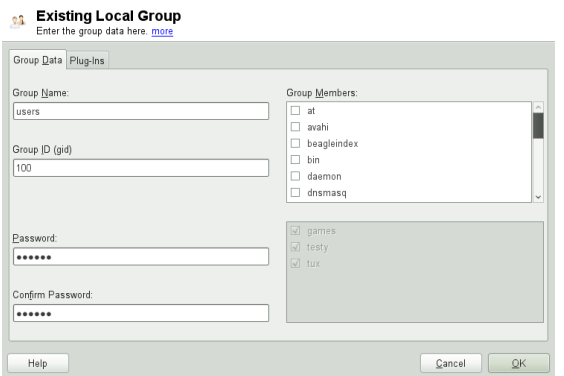
1 Open the YaST *User and Group Management* dialog and click the *Groups* tab.

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 Chapitre 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 Chapitre 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 » (Chapitre 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 Chapitre 19, *Samba* (↑Reference) and Chapitre 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 configu-

ration modules in YaST. For information about the configuration of the appropriate client, refer to the following sections:

NIS: section « Configuring NIS Clients » (Chapitre 3, *Using NIS*, ↑Security Guide)

LDAP: section « Configuring an LDAP Client with YaST » (Chapitre 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 » (p. 137). 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 » (p. 140).

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 » (p. 138) and Section 11.1.2, « Switching the Default System Language » (p. 139) 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 » (p. 140).

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.



Procédure 11.1 *Installing Additional Languages*

When installing additional languages, YaST also allows you to set different locale settings for the user `root`, see Étape 4 (p. 139). 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.

ASTUCE

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 » (p. 140).

---

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 » (p. 140).

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.

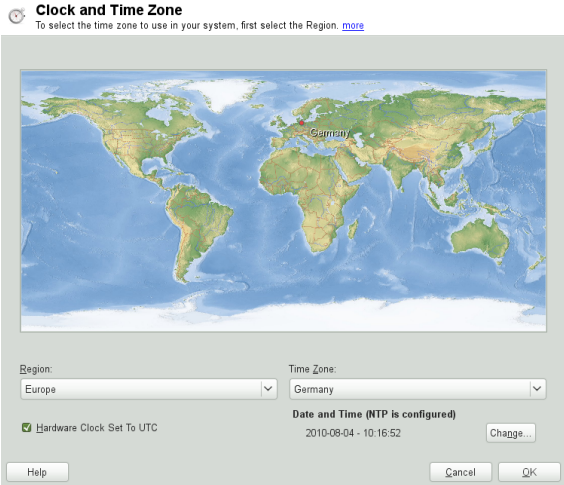
KDE Applications
: Start the application from the command line by using the following command:

  `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.

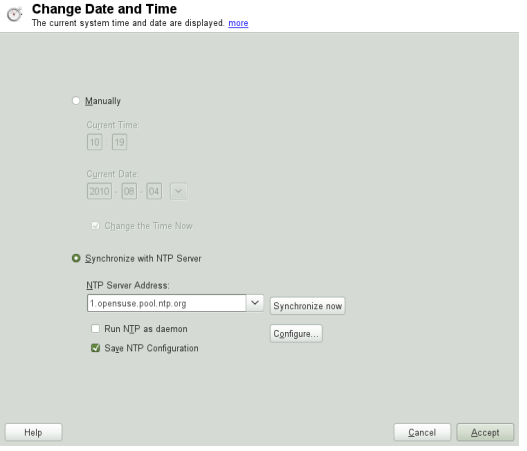
Procédure 11.2 *Manually Adjusting Time and Date*

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*.

Procédure 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*.

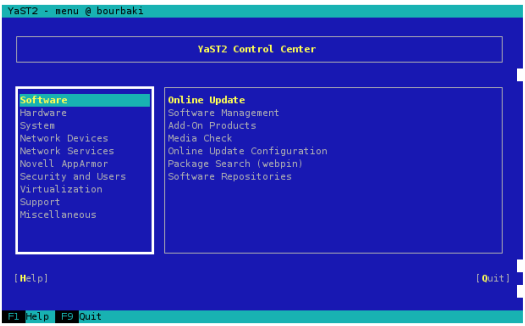
6 With the *Configure* button, you can open the advanced NTP configuration. For details, see section « Configuring an NTP Client with YaST » (Chapitre 17, *Time Synchronization with NTP*, ↑Reference).

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 [Entrée] to start the active module.

Various buttons or selection fields in the module contain a highlighted letter (yellow by default). Use [Alt] + [highlighted_letter] to select a button directly instead of navigating there with [Tab]. Exit the YaST Control Center by pressing [Alt] + [Q] or by selecting *Quit* and pressing [Entrée].

12.1 Navigation in Modules

The following description of the control elements in the YaST modules assumes that all function keys and [Alt] key combinations work and are not assigned to different global functions. Read Section 12.2, « Restriction of Key Combinations » (p. 145) 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 [Alt] + [Tab] or [Maj] + [Tab] combinations.

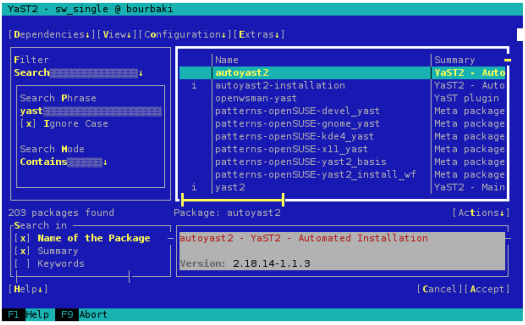
Navigation in Selection Lists
: Use the arrow keys ([↑] and [↓]) to navigate among the individual elements in an active frame containing a selection list. If individual entries within a frame exceed its width, use [Maj] + [→] or [Maj] + [←] to scroll horizontally to the right and left. Alternatively, use [Ctrl] + [E] or [Ctrl] + [A]. This combination can also be used if using [→] or [←] results in changing the active frame or the current selection list, as in the Control Center.

Buttons, Radio Buttons, and Check Boxes
: To select buttons with empty square brackets (check boxes) or empty parentheses (radio buttons), press [Space] or [Entrée]. Alternatively, radio buttons and check boxes can be selected directly with [Alt] + [highlighted_letter]. In this case, you do not need to confirm with [Entrée]. If you navigate to an item with [Tab], press [Entrée] 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, [Entrée] 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 [Alt] combinations, the [Alt] combinations in YaST might not work. Keys like [Alt] or [Maj] can also be occupied by the settings of the terminal.

Replacing [Alt] with [Esc]
: [Alt] shortcuts can be executed with [Esc] instead of [Alt]. For example, [Esc] – [H] replaces [Alt] + [H]. (First press [Esc], *then* press [H].)

Backward and Forward Navigation with [Ctrl] + [F] and [Ctrl] + [B]
: If the [Alt] and [Maj] combinations are occupied by the window manager or the terminal, use the combinations [Ctrl] + [F] (forward) and [Ctrl] + [B] (backward) instead.

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

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 » (p. 105).

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.

ASTUCE : 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.

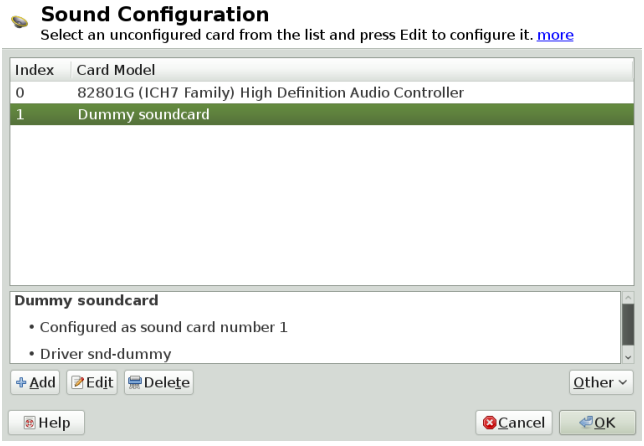
3. Click *Save to File...* to save the output to a file.

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.

Procédure 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.

Procédure 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.

   REMARQUE : 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 » (p. 150), instructions on how to configure a network scanner are available at Section 13.3.3, « Scanning over the Network » (p. 151).

Procédure 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 Procédure 13.3, « Configuring a USB or SCSI Scanner » (p. 150). 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 » (p. 150).
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 » (p. 155). 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 » (p. 164).

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 » (p. 164) 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/collaborate/workgroups/openprinting/database/databaseintro`
: The OpenPrinting home page with the printer database. The database shows the latest Linux support status. However, a Linux distribution can only integrate

the drivers available at production time. Accordingly, a printer currently rated as « perfectly supported » may not have had this status when the latest openSUSE version was released. Thus, the database may not necessarily indicate the correct status, but only provide an approximation.

`http://pages.cs.wisc.edu/~ghost/`
The Ghostscript Web page.

`/usr/share/doc/packages/ghostscript/catalog.devices`
List of 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`.

AVERTISSEMENT : 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 of a printer, 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 » (p. 162) and Section 14.8.2, « No Suitable PPD File Available for a PostScript Printer » (p. 164).

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 » (p. 157). 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.

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: Specify the connection type, 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, 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 » (p. 157) 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 » (p. 154) for more information about print queues.

Procédure 14.1 *Adding a New Local Printer*

1. Start the YaST printer module with *Hardware* > *Printer*.
2. In the *Printer Configurations* screen click *Add*.
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. In the input box under `Find and Assign a Driver` enter the vendor name and the model name 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 » (p. 157)
6. Specify the `Default paper size`.
7. In the *Set Arbitrary Name* field enter a unique name for the printer queue.
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 list of printers.

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 » (p. 155).

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 » (p. 164). Alternatively, find PPD files at `http://www.linuxfoundation.org/collaborate/workgroups/openprinting/database/databaseintro`, the « OpenPrinting.org printer database ». When downloading PPD files from OpenPrinting, keep in mind that it always shows the latest Linux support status, which is not necessarily met by openSUSE.

Procédure 14.2 *Adding a PPD file*

1. Start the YaST printer module with *Hardware* > *Printer*.
2. In the *Printer Configurations* screen click *Add*.
3. In the `Find and Assign a Driver` section click *Driver Packages*.
4. Enter the full path to the PPD file into the input box under `Make a Printer Description File Available`. Alternatively, choose the file from a dialog box by clicking *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 Procédure 14.1, « Adding a New Local Printer » (p. 156). 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.

Procédure 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 Procédure 14.1, « Adding a New Local Printer » (p. 156). 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 an option. Apply your changes with *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.

Procédure 14.4 *Printing via a Single CUPS Server*

1 Start the YaST printer module with *Hardware* > *Printer*.

2 From the left pane launch the *Print via Network* screen.

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.

Procédure 14.5 *Printing via Multiple CUPS Servers*

1 Start the YaST printer module with *Hardware* > *Printer*.

2 From the left pane launch the *Print via Network* screen.

3 Check *Accept Printer Announcements from CUPS Servers*.

4 Under `General Settings` specify which servers to use. 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.

5 Confirm by clicking *OK* and then *Yes* when asked to start a local CUPS server. After the server has started YaST 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.

Procédure 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.
   - 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 to 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 para-

meters. 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 » (Chapitre 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 confi-

guration generates a database with 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`, but 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.

The PPD file is provided as a zip archive (.zip) or a self-extracting zip archive (`.exe`). Unpack the zip archive 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 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 Exemple 14.1, « Error Message from `lpd` » (p. 165), the problem is caused by the remote `lpd`.

Exemple 14.1 *Error Message from lpd*

```
lpd: your host does not have line printer access
lpd: queue does not exist
printer: spooling disabled
printer: printing disabled
```

Checking a Remote `cupsd`

A CUPS network server can broadcast its queues by default every 30 seconds on UDP port `631`. Accordingly, the following command can be used to test whether there is a broadcasting CUPS network server in the network. Make sure to stop your local CUPS daemon before executing the command.

```
netcat -u -l -p 631 & PID=$! ; sleep 40 ; kill $PID
```

If a broadcasting CUPS network server exists, the output appears as shown in Exemple 14.2, « Broadcast from the CUPS Network Server » (p. 166).

Exemple 14.2 *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 » (p. 159).

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
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.

14.8.4 Defective Printouts without Error Message

For the print system, the print job is completed when the CUPS back-end completes the data transfer to the recipient (printer). If further processing on the recipient fails (for example, if the printer is not able to print the printer-specific data) the print system does not notice this. If the printer is not able to print the printer-specific data, select a PPD file that is more suitable for the printer.

14.8.5 Disabled Queues

If the data transfer to the recipient fails entirely after several attempts, the CUPS back-end, such as `USB` or `socket`, reports an error to the print system (to `cupsd`). The back-end determines how many unsuccessful attempts are appropriate until the data transfer is reported as impossible. As further attempts would be in vain, `cupsd` disables printing for the respective queue. After eliminating the cause of the problem, the system administrator must reenable printing with the command `cupsenable`.

14.8.6 CUPS Browsing: Deleting Print Jobs

If a CUPS network server broadcasts its queues to the client hosts via browsing and a suitable local `cupsd` is active on the client hosts, the client `cupsd` accepts print jobs from applications and forwards them to the `cupsd` on the server. When `cupsd` on the server accepts a print job, it is assigned a new job number. Therefore, the job number on the client host is different from the job number on the server. As a print job is usually forwarded immediately, it cannot be deleted with the job number on the client host This is because the client `cupsd` regards the print job as completed 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`*

14.8.7 Defective Print Jobs and Data Transfer Errors

If you switch the printer off or shut down the computer during the printing process, print jobs remain in the queue. Printing resumes when the computer (or the printer) is switched back on. Defective print jobs must be removed from the queue with `cancel`.

If a print job is defective or an error occurs in the communication between the host and the printer, the printer prints numerous sheets of paper with unintelligible characters, because it is unable to process the data correctly. To rectify this situation, follow these steps:

1. To stop printing, remove all paper from the printer or open its paper trays. 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.

14.8.8 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.

14.8.9 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 enable the 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 Chapitre 5, *Installation et Désinstallation de Logiciels* (p. 75) and Chapitre 6, *YaST Online Update* (p. 95). Two ways to upgrade the system are discussed in the following sections — see Section 16.1.3, « Upgrading with YaST » (p. 172) and Section 16.1.4, « Distribution Upgrade with zypper » (p. 173).

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 Exemple 16.1, « List with `df -h` » (p. 171), the root partition to write down is `/dev/sda3` (mounted as `/`).

Exemple 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 » (p. 171), you can now upgrade your system:

1. Boot the system as for the installation, described in section « Démarrage du système pour l'installation » (Chapitre 1, *Installation avec 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 « Produits Additionnels » (Chapitre 1, *Installation avec 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 » (p. 171)). YaST reads the old `fstab` on this partition to analyze and mount the file systems listed there.

   AVERTISSEMENT : 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

AVERTISSEMENT : 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 Chapitre 6, *YaST Online Update* (p. 95).

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 » (p. 86)) or `zypper` (see Section 9.1, « Using Zypper » (p. 105)). 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
```

ASTUCE : 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.

REMARQUE : 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 Chapitre 5, *Installation et Désinstallation de Logiciels* (p. 75). 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 Chapitre 6, *YaST Online Update* (p. 95).

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.

Partie 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 Chapitre 18, *Bash and Bash Scripts* (p. 211).

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 [Ctrl] + [Alt] + [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 [Entrée]. Then type your password and press [Entrée]. 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 [Entrée]. Then press [Alt] + [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 [Alt] + [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 [Alt] + [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: [←], [→], [Deébut], [End], [<—] (Backspace), [Suppr], and [Space]. You can correct typing errors or add options. The command is not executed until you press [Entrée].

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 » (p. 188) 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 » (p. 188).
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 » (p. 188).
❹ Group assigned to the file or directory. For further information, refer to Section 17.5.1, « Permissions for User, Group and Others » (p. 188).
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 [Page ↑] and [Page ↓]. Move between the beginning and the end of a document with [Deébut] and [End]. 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 [Space] to move forward and [<—] to move backwards. Within a node, you can also browse with [Page ↑] and [Page ↓] but only [Space] and [<—] will take you also to the previous or subsequent node. 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
[Ctrl] + [L]	Clears the screen and moves the current line to the top of the page.
[Ctrl] + [C]	Aborts the command which is currently being executed.
[Maj] + [Page ↑]	Scrolls upwards.
[Maj] + [Page ↓]	Scrolls downwards.
[Ctrl] + [U]	Deletes from cursor position to start of line.
[Ctrl] + [K]	Deletes from cursor position to the end of line.
[Ctrl] + [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.

REMARQUE : 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 » (p. 188)). 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.

Procédure 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
```

Procédure 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`.

Procédure 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 `root`s 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 Chapitre 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.

Exemple 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 Exemple 17.1, « Access Permissions For Files and Folders » (p. 188).

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:

Tableau 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	Users can create, rename or delete files in the directory.

Access Permission	File	Folder
	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.	
Execute (x)	Users can execute the file. This permission is only relevant for files like programs or shell 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.	Users can change into the directory and execute files there. If they do not have read access to 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 » (p. 188).

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.

Exemple 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.

Procédure 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
```

Procédure 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 [Page ↑].

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 [Entrée]. 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 [Ctrl] + [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 [Esc]. 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 [Entrée].

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.

Procédure 17.6 *Using History and Completion*

If you already did the example Section 17.3.1, « Examples for Working with Files and Directories » (p. 184) your shell buffer should be filled with commands which you can retrieve using the history function.

1 Press [↑] repeatedly until `cd ~` appears.

2 Press [Entrée] 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 [Entrée].

Procédure 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.

REMARQUE : 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

REMARQUE : 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 [Maj] 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 [Esc] 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 [Suppr].

5. To leave vi, press [Esc] 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 [Entrée]. If you want to save the file under a different name, type `w` *`filename`* and press [Entrée].

   To leave vi without saving, type `q!` instead and press [Entrée].

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 [Page ↑] and [Page ↓]. Use [Space] to scroll down one page. [Deébut] takes you to the beginning, and [End] 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 [Space] 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 » (p. 196), 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 [Ctrl] + [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 » (Chapitre 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 [Page ↑] and [Page ↓]. Move between the beginning and the end of a document with [Deébut] and [End]. 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 » (Chapitre 11, *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 automated tasks. Shells are a good addition to GUIs and this chapter gives you an overview of some aspects of shells, in this case Bash.

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.

Tableau 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

Tableau 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:

Tableau 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.

Tableau 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`.

REMARQUE : 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 Tableau A.2, « Log Files » (p. 228).

`/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 » (Chapitre 27, *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:

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

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

character. For example, the following line searches for a file starting with `foo`, but suppresses its errors by redirecting it to `/dev/null`:

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

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:

Tableau 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 » (p. 223).

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 » (p. 225) and Section A.1.3, « Info Pages » (p. 226).

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.

REMARQUE : 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 [Page ↑] and [Page ↓]. Move between the beginning and the end of a document with [Deébut] and [End]. End this viewing mode by pressing [Q]. Learn more about the `man` command itself with `man man`. Man pages are sorted in categories as shown in Tableau A.1, « Man Pages—Categories and Descriptions » (p. 225) (taken from the man page for man itself).

Tableau 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`)

Number	Description
6	Games
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 [Space] to move forward and [<—] to move backwards. Within a node, you can also browse with [Page ↑] and [Page ↓] but only [Space] and [<—] will take you also to the previous or subsequent node. Press [Q] to end the viewing mode. Not every 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 like Google™ [http://www.google.com/linux]. 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.

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.

Tableau A.2 *Log Files*

Log File	Description
`~/.xsession-errors`	Messages from the desktop applications currently running.
`/var/log/apparmor/`	Log files from AppArmor, see Partie « 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.

Log File	Description
/var/log/YaST2/	Directory containing YaST's actions and their results.
/var/log/zypper.log	Log file of zypper.

Apart from log files, your machine also supplies you with information about the running system. See Tableau A.3: System Information With the /proc File System

Tableau 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 Chapitre 12, *Dynamic Kernel Device Management with* `udev` (↑Reference). Tableau A.4 contains an overview of the most common directories under `/sys`.

Tableau 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 Chapitre 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 « Installation à Distance via VNC—PXE Démarrage et Réveil par le LAN (Réseau Local) » (Chapitre 2, *Installation à distance*, ↑Reference) and section « Installation à distance via SSH—démarrage PXE et réveil par le LAN » (Chapitre 2, *Installation à distance*, ↑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.

Procédure 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].

Procédure 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 [Ctrl] + [Alt] + [Suppr] 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 [Entrée] 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 Chapitre 1, *Installation avec 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`.

ASTUCE : 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.

Procédure 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 Chapitre 1, *Installation avec YaST* (↑Reference).

Procédure 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 Chapitre 1, *Installation avec YaST* (↑Reference).

Procédure 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 [Entrée] 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 [Entrée]:

   ```
   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 Chapitre 1, *Installation avec 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 [Entrée] 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 » (p. 233). To launch the installation process, press [Entrée].

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 » (p. 249) and follow the instructions at la section intitulée « Modifying and Reinstalling the Boot Loader » (p. 252).

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.

ASTUCE : 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 » (p. 243).
- 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 » (p. 243).
- 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 Chapitre 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 » (p. 241).

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 [Ctrl] + [Alt] + [F1]). If this is successful, the blame cannot be put on PAM, because it is possible to authenticate this user on this machine. Try to locate any problems with the X Window System or the desktop (GNOME or KDE). For more information, refer to Section A.2.4.4, « Login Successful but GNOME Desktop Fails » (p. 241) and Section A.2.4.5, « Login Successful but KDE Desktop Fails » (p. 242).
4. If the user's home directory has been used with another Linux distribution, remove the `Xauthority` file in the user's home. Use a console login via [Ctrl] + [Alt] + [F1] and run `rm .Xauthority` as this user. This should eliminate X authentication problems for this user. Try graphical login again.
5. If graphical login still fails, do a console login with [Ctrl] + [Alt] + [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 » (p. 241) or Section A.2.4.5, « Login Successful but KDE Desktop Fails » (p. 242).

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 [Ctrl] + [Alt] + [F1]). If this is successful, the problem is not with PAM or the directory server on which the user's home is hosted, because it is possible to authenticate this user on this machine. Try to locate any problems with the X Window System or the desktop (GNOME or KDE). For more information, refer to Section A.2.4.4, « Login Successful but GNOME Desktop Fails » (p. 241) and Section A.2.4.5, « Login Successful but KDE Desktop Fails » (p. 242).
6. If the user's home directory has been used with another Linux distribution, remove the `Xauthority` file in the user's home. Use a console login via [Ctrl] + [Alt] + [F1] and run `rm .Xauthority` as this user. This should eliminate X authentication problems for this user. Try graphical login again.
7. If graphical login still fails, do a console login with [Ctrl] + [Alt] + [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 » (p. 241) or Section A.2.4.5, « Login Successful but KDE Desktop Fails » (p. 242).

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 [Ctrl] + [Alt] + [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 [Alt] + [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 [Ctrl] + [Alt] + [F1].

2 Log in with your user name.

3 Move the user's GNOME configuration directories to a temporary location:

```
mv .gconf  .gconf-ORIG-RECOVER
mv .gnome2 .gnome2-ORIG-RECOVER
```

4 Log out.

5 Log in again, but do not run any applications.

6 Recover your individual application configuration data (including the Evolution e-mail client data) by copying the `~/.gconf-ORIG-RECOVER/apps/` directory back into the new `~/.gconf` directory as follows:

```
cp -a .gconf-ORIG-RECOVER/apps .gconf/
```

If this causes the login problems, attempt to recover only the critical application data and reconfigure the remainder of the applications.

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 [Ctrl] + [Alt] + [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 Étape 3 (p. 242).

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.

Procédure 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 Étape 4b (p. 245). 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 Chapitre 15, *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 Procédure A.6, « How to Identify Network Problems » (p. 243). 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 Chapitre 25, *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.

Procédure 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 « Réveil par le Réseau Local (Wake on LAN) » (Chapitre 2, *Installation à distance*, ↑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 [Alt] + [F1] to [Alt] + [F6].

A shell and many other useful utilities, such as the mount program, are available in the `/bin` directory. The `sbin` directory contains important file and network utilities for reviewing and repairing the file system. This directory also contains the most important binaries for system maintenance, such as fdisk, mkfs, mkswap, mount, 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`.

AVERTISSEMENT : 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 [Alt] + [F1] to [Alt] + [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 la section intitulée « Accessing the Installed System » (p. 251).

2 Check whether the following files are correctly configured according to the GRUB configuration principles outlined in Chapitre 9, *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.

ASTUCE : 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 la section intitulée « Accessing the Installed System » (p. 251).
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 la section intitulée « Modifying and Reinstalling the Boot Loader » (p. 252)

8 Remove any bootable media from the system drive and reboot.

A.3 Legal Notice

Copyright© 2006–2014 SUSE LLC and contributors. All rights reserved.

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

For SUSE or 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 SUSE or 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 SUSE LLC, SUSE LINUX Products GmbH, the authors, nor the translators shall be held liable for possible errors or the consequences thereof.

GNU Free Documentation License

Copyright (C) 2000, 2001, 2002 Free Software Foundation, Inc. 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA. Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

0. PREAMBLE

The purpose of this License is to make a manual, textbook, or other functional and useful document "free" in the sense of freedom: to assure everyone the effective freedom to copy and redistribute it, with or without modifying it, either commercially or 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.

1. APPLICABILITY AND DEFINITIONS

This License applies to any manual or other work, in any medium, that contains a notice placed by the copyright holder saying it can be distributed under the terms of this License. Such a notice grants a world-wide, royalty-free license, unlimited in duration, to use that work under the conditions stated herein. The "Document", below, refers to any such manual or work. Any member of the public is a licensee, and is addressed as "you". You accept the license if you copy, modify or distribute the work in a way requiring permission under copyright law.

A "Modified Version" of the Document means any work containing the Document or a portion of it, either copied verbatim, or with modifications and/or translated into another language.

A "Secondary Section" is a named appendix or a front-matter section of the Document that deals exclusively with the relationship of the publishers or authors of the Document to the Document's overall subject (or to related matters) and contains nothing that could fall directly within that overall subject. (Thus, if the Document is in part a textbook of mathematics, a Secondary Section may not explain any mathematics.) The relationship could be a matter of historical connection with the subject or with related matters, or of legal, commercial, philosophical, ethical or political position regarding them.

The "Invariant Sections" are certain Secondary Sections whose titles are designated, as being those of Invariant Sections, in the notice that says that the Document is released under this License. If a section does not fit the above definition of Secondary then it is not allowed to be designated as Invariant. The Document may contain zero Invariant Sections. If the Document does not identify any Invariant Sections then there are none.

The "Cover Texts" are certain short passages of text that are listed, as Front-Cover Texts or Back-Cover Texts, in the notice that says that the Document is released under this License. A Front-Cover Text may be at most 5 words, and a Back-Cover Text may be at most 25 words.

A "Transparent" copy of the Document means a machine-readable copy, represented in a format whose specification is available to the general public, that is suitable for revising the document straightforwardly with generic text editors or (for images composed of pixels) generic paint programs or (for drawings) some widely available 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.

2. 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.

3. COPYING IN QUANTITY

If you publish printed copies (or copies in media that commonly have printed covers) of the Document, numbering more than 100, and the Document's license notice requires Cover Texts, you must enclose the copies in covers that carry, clearly and legibly, all these Cover Texts: Front-Cover Texts on the front cover, and Back-Cover Texts on the back cover. Both covers must also clearly and legibly identify you as the publisher of these copies. The front cover must present the full title with all words of the title equally prominent and visible. You may add other material on the covers in addition. Copying with changes limited to the covers, as long as they preserve the title of the Document and satisfy these conditions, can be treated as verbatim copying in other respects.

If the required texts for either cover are too voluminous to fit legibly, you should put the first ones listed (as many as fit reasonably) on the actual cover, and continue the rest onto adjacent pages.

If you publish or distribute Opaque copies of the Document numbering more than 100, you must either include a machine-readable Transparent copy along with each Opaque copy, or state in or with each Opaque copy a computer-network location from which the general network-using public has access to download using public-standard network protocols a complete Transparent copy of the Document, free of added material. If you use the latter option, you must take reasonably prudent steps, when you begin distribution of Opaque copies in quantity, to ensure that this Transparent copy will

remain thus accessible at the stated location until at least one year after the last time you distribute an Opaque copy (directly or through your agents or retailers) of that edition to the public.

It is requested, but not required, that you contact the authors of the Document well before redistributing any large number of copies, to give them a chance to provide you with an updated version of the Document.

4. MODIFICATIONS

You may copy and distribute a Modified Version of the Document under the conditions of sections 2 and 3 above, provided that you release the Modified Version under precisely this License, with the Modified Version filling the role of the Document, thus licensing distribution and modification of the Modified Version to whoever possesses a copy of it. In addition, you must do these things in the Modified Version:

A. Use in the Title Page (and on the covers, if any) a title distinct from that of the Document, and from those of previous versions (which should, if there were any, be listed in the History section of the Document). You may use the same title as a previous version if the original publisher of that version gives permission.

B. List on the Title Page, as authors, one or more persons or entities responsible for authorship of the modifications in the Modified Version, together with at least five of the principal authors of the Document (all of its principal authors, if it has fewer than five), unless they release you from this requirement.

C. State on the Title page the name of the publisher of the Modified Version, as the publisher.

D. Preserve all the copyright notices of the Document.

E. Add an appropriate copyright notice for your modifications adjacent to the other copyright notices.

F. Include, immediately after the copyright notices, a license notice giving the public permission to use the Modified Version under the terms of this License, in the form shown in the Addendum below.

G. Preserve in that license notice the full lists of Invariant Sections and required Cover Texts given in the Document's license notice.

H. Include an unaltered copy of this License.

I. Preserve the section Entitled "History", Preserve its Title, and add to it an item stating at least the title, year, new authors, and publisher of the Modified Version as given on the Title Page. If there is no section Entitled "History" in the Document, create one stating the title, year, authors, and publisher of the Document as given on its Title Page, then add an item describing the Modified Version as stated in the previous sentence.

J. Preserve the network location, if any, given in the Document for public access to a Transparent copy of the Document, and likewise the network locations given in the Document for previous versions it was based on. These may be placed in the "History" section. You may omit a network location for a work that was published at least four years before the Document itself, or if the original publisher of the version it refers to gives permission.

K. For any section Entitled "Acknowledgements" or "Dedications", Preserve the Title of the section, and preserve in the section all the substance and tone of each of the contributor acknowledgements and/or dedications given therein.

L. Preserve all the Invariant Sections of the Document, unaltered in their text and in their titles. Section numbers or the equivalent are not considered part of the section titles.

M. Delete any section Entitled "Endorsements". Such a section may not be included in the Modified Version.

N. Do not retitle any existing section to be Entitled "Endorsements" or to conflict in title with any Invariant Section.

O. Preserve any Warranty Disclaimers.

If the Modified Version includes new front-matter sections or appendices that qualify as Secondary Sections and contain no material copied from the Document, you may at your option designate some or all of these sections as invariant. To do this, add their titles to the list of Invariant Sections in the Modified Version's license notice. These titles must be distinct from any other section titles.

You may add a section Entitled "Endorsements", provided it contains nothing but endorsements of your Modified Version by various parties--for example, statements of peer review or that the text has been approved by an organization as the authoritative definition of a standard.

You may add a passage of up to five words as a Front-Cover Text, and a passage of up to 25 words as a Back-Cover Text, to the end of the list of Cover Texts in the Modified Version. Only one passage of Front-Cover Text and one of Back-Cover Text may be added by (or through arrangements made by) any one entity. If the Document already includes a cover text for the same cover, previously added by you or by arrangement made by the same entity you are acting on behalf of, you may not add another; but you may replace the old one, on explicit permission from the previous publisher that added the old one.

The author(s) and publisher(s) of the Document do not by this License give permission to use their names for publicity for or to assert or imply endorsement of any Modified Version.

5. COMBINING DOCUMENTS

You may combine the Document with other documents released under this License, under the terms defined in section 4 above for modified versions, provided that you include in the combination all of the Invariant Sections of all of the original documents, unmodified, and list them all as Invariant Sections of your combined work in its license notice, and that you preserve all their Warranty Disclaimers.

The combined work need only contain one copy of this License, and multiple identical Invariant Sections may be replaced with a single copy. If there are multiple Invariant Sections with the same name but different contents, make the title of each such section unique by adding at the end of it, in parentheses, the name of the original author or publisher of that section if known, or else a unique number. Make the same adjustment to the section titles in the list of Invariant Sections in the license notice of the combined work.

In the combination, you must combine any sections Entitled "History" in the various original documents, forming one section Entitled "History"; likewise combine any sections Entitled "Acknowledgements", and any sections Entitled "Dedications". You must delete all sections Entitled "Endorsements".

6. COLLECTIONS OF DOCUMENTS

You may make a collection consisting of the Document and other documents released under this License, and replace the individual copies of this License in the various documents with a single copy that is included in the collection, provided that you follow the rules of this License for verbatim copying of each of the documents in all other respects.

You may extract a single document from such a collection, and distribute it individually under this License, provided you insert a copy of this License into the extracted document, and follow this License in all other respects regarding verbatim copying of that document.

7. AGGREGATION WITH INDEPENDENT WORKS

A compilation of the Document or its derivatives with other separate and independent documents or works, in or on a volume of a storage or distribution medium, is called an "aggregate" if the copyright resulting from the compilation is not used to limit the legal rights of the compilation's users beyond what the individual works permit. When the Document is included in an aggregate, this License does not apply to the other works in the aggregate which are not themselves derivative works of the Document.

If the Cover Text requirement of section 3 is applicable to these copies of the Document, then if the Document is less than one half of the entire aggregate, the Document's Cover Texts may be placed on covers that bracket the Document within the aggregate, or the electronic equivalent of covers if the Document is in electronic form. Otherwise they must appear on printed covers that bracket the whole aggregate.

8. TRANSLATION

Translation is considered a kind of modification, so you may distribute translations of the Document under the terms of section 4. Replacing Invariant Sections with translations requires special permission from their copyright holders, but you may include translations of some or all Invariant Sections in addition to the original versions of these Invariant Sections. You may include a translation of this License, and all the license notices in the Document, and any Warranty Disclaimers, provided that you also include the original English version of this License and the original versions of those notices and disclaimers. In case of a disagreement between the translation and the original version of this License or a notice or disclaimer, the original version will prevail.

If a section in the Document is Entitled "Acknowledgements", "Dedications", or "History", the requirement (section 4) to Preserve its Title (section 1) will typically require changing the actual title.

9. TERMINATION

You may not copy, modify, sublicense, or distribute the Document except as expressly provided for under this License. Any other attempt to copy, modify, sublicense or distribute the Document is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

10. FUTURE REVISIONS OF THIS LICENSE

The Free Software Foundation may publish new, revised versions of the GNU Free Documentation License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns. See http://www.gnu.org/copyleft/.

Each version of the License is given a distinguishing version number. If the Document specifies that a particular numbered version of this License "or any later version" applies to it, you have the option of following the terms and conditions either of that specified version or of any later version that has been published (not as a draft) by the Free Software Foundation. If the Document does not specify a version number of this License, you may choose any version ever published (not as a draft) by the Free Software Foundation.

ADDENDUM: How to use this License for your documents

```
Copyright (c) YEAR YOUR NAME.
Permission is granted to copy, distribute and/or modify this document
under the terms of the GNU Free Documentation License, Version 1.2
or any later version published by the Free Software Foundation;
with no Invariant Sections, no Front-Cover Texts, and no Back-Cover Texts.
A copy of the license is included in the section entitled “GNU
Free Documentation License”.
```

If you have Invariant Sections, Front-Cover Texts and Back-Cover Texts, replace the “with...Texts.” line with this:

```
    with the Invariant Sections being LIST THEIR TITLES, with the
    Front-Cover Texts being LIST, and with the Back-Cover Texts being LIST.
```

If you have Invariant Sections without Cover Texts, or some other combination of the three, merge those two alternatives to suit the situation.

If your document contains nontrivial examples of program code, we recommend releasing these examples in parallel under your choice of free software license, such as the GNU General Public License, to permit their use in free software.