

openSUSE

12.3

www.suse.com

Novembre 06, 2013

Reference



Reference

Copyright © 2006–2013 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

À propos de ce guide **xiii**

1 Documentation disponible	xiv
2 Remarques	xv
3 Conventions de la documentation	xv
4 À propos de la création de ce manuel	xvi
5 Code source	xvi
6 Remerciements	xvi

I Installation et Déploiement **1**

1 Installation avec YaST **3**

1.1 Choix du Média d'installation	3
1.2 Choisir la méthode d'installation	5
1.3 L'Automatisation des Processus (Workflow) d'Installation	9
1.4 Démarrage du système pour l'installation	9
1.5 l'Ecran de Démarrage	10
1.6 Bienvenue	13
1.7 Mode installation	14
1.8 Heure et fuseaux horaires.	17
1.9 Sélection du bureau	18
1.10 Partitionnement suggéré	19
1.11 Créez un nouvel utilisateur	22

1.12 Paramètres d'Installation	26
1.13 Exécution de l'installation	30
1.14 Configuration du système installé.	31
1.15 Login Graphique	36

2 Installation à distance 37

2.1 Scénarios d'Installation à Distance	37
2.2 Configuration du serveur contenant les sources de l'installation.	46
2.3 Préparer le démarrage du système cible	56
2.4 Démarrage du Système Cible pour l'Installation	66
2.5 Surveillance du Processus d'Installation	70

3 Configuration avancée des disques 75

3.1 Utilisation du partitionneur YaST	75
3.2 Configuration LVM	86
3.3 Soft RAID Configuration	92

II Advanced Administration 97

4 Snapshots/Rollback with Snapper 99

4.1 Requirements	99
4.2 Using Snapper to Undo YaST and zypper Changes	100
4.3 Using Snapper to Restore Files from Hourly Backups	106
4.4 Managing Snapshots	106
4.5 Disabling Snapper	109
4.6 Creating Snapper Configurations	110
4.7 Limitations	113
4.8 Frequently Asked Questions	114

5 Remote Access with VNC 115

5.1 One-time VNC Sessions	115
5.2 Persistent VNC Sessions	118

III Système 121

6 Applications 32 bits et 64 bits dans un environnement système 64 bits 123

6.1 Support d'exécution	123
6.2 Développement logiciel	124
6.3 Compilation logicielle sur plateformes biarch	125
6.4 Spécifications du noyau	126

7 Booting a Linux System 127

7.1 The Linux Boot Process	127
----------------------------------	-----

8 The `systemd` daemon 133

8.1 Basic Usage	134
8.2 System Start and Target Management	141
8.3 Advanced Usage	148
8.4 More information	151

9 The Boot Loader GRUB 153

9.1 Booting with GRUB	154
9.2 Configuring the Boot Loader with YaST	164
9.3 Uninstalling the Linux Boot Loader	169
9.4 Creating Boot CDs	170
9.5 The Graphical SUSE Screen	171
9.6 Troubleshooting	172
9.7 For More Information	173

10 The Boot Loader GRUB2	175
10.1 Main Differences from GRUB Legacy	175
10.2 Configuration File Structure	176
10.3 Configuring the Boot Loader with YaST	184
10.4 For More Information	189
11 Special System Features	191
11.1 Information about Special Software Packages	191
11.2 Virtual Consoles	198
11.3 Keyboard Mapping	198
11.4 Language and Country-Specific Settings	199
12 Dynamic Kernel Device Management with udev	203
12.1 The /dev Directory	203
12.2 Kernel uevents and udev	204
12.3 Drivers, Kernel Modules and Devices	204
12.4 Booting and Initial Device Setup	205
12.5 Monitoring the Running udev Daemon	205
12.6 Influencing Kernel Device Event Handling with udev Rules	207
12.7 Persistent Device Naming	213
12.8 Files used by udev	214
12.9 For More Information	214
IV Services	217
13 Basic Networking	219
13.1 IP Addresses and Routing	222
13.2 IPv6—The Next Generation Internet	225
13.3 Name Resolution	235

13.4 Configuring a Network Connection with YaST	236
13.5 NetworkManager	257
13.6 Configuring a Network Connection Manually	260
13.7 smpppd as Dial-up Assistant	275

14 SLP Services in the Network 279

14.1 Installation	279
14.2 Activating SLP	280
14.3 SLP Front-Ends in openSUSE	280
14.4 Installation over SLP	280
14.5 Providing Services via SLP	281
14.6 For More Information	282

15 The Domain Name System 283

15.1 DNS Terminology	283
15.2 Installation	284
15.3 Configuration with YaST	284
15.4 Starting the BIND Name Server	294
15.5 The /etc/named.conf Configuration File	296
15.6 Zone Files	300
15.7 Dynamic Update of Zone Data	304
15.8 Secure Transactions	304
15.9 DNS Security	306
15.10 For More Information	306

16 DHCP 307

16.1 Configuring a DHCP Server with YaST	308
16.2 DHCP Software Packages	311
16.3 The DHCP Server dhcpd	312

16.4 For More Information	315
17 Time Synchronization with NTP	317
17.1 Configuring an NTP Client with YaST	317
17.2 Manually Configuring ntp in the Network	322
17.3 Dynamic Time Synchronization at Runtime	322
17.4 Setting Up a Local Reference Clock	323
18 Sharing File Systems with NFS	325
18.1 Terminology	325
18.2 Installing NFS Server	326
18.3 Configuring NFS Server	326
18.4 Configuring Clients	335
18.5 For More Information	338
19 Samba	339
19.1 Terminology	339
19.2 Installing a Samba Server	341
19.3 Starting and Stopping Samba	341
19.4 Configuring a Samba Server	341
19.5 Configuring Clients	348
19.6 Samba as Login Server	348
19.7 For More Information	349
20 Le serveur HTTP Apache	351
20.1 Démarrage rapide	351
20.2 Configuration d'Apache	353
20.3 Démarrage et arrêt d'Apache	368
20.4 Installation, activation et configuration de modules	371
20.5 Getting CGI Scripts to Work	379

20.6 Setting Up a Secure Web Server with SSL	381
20.7 Avoiding Security Problems	388
20.8 Troubleshooting	390
20.9 For More Information	391

21 Setting up an FTP server with YaST **393**

V Mobilité **397**

22 Mobile Computing with Linux **399**

22.1 Laptops	399
22.2 Mobile Hardware	406
22.3 Cellular Phones and PDAs	407
22.4 For More Information	407

23 Power Management **409**

23.1 Power Saving Functions	409
23.2 Advanced Configuration and Power Interface (ACPI)	410
23.3 Rest for the Hard Disk	413
23.4 Troubleshooting	415
23.5 For More Information	416

24 Wireless LAN **419**

24.1 WLAN Standards	419
24.2 Operating Modes	420
24.3 Authentication	421
24.4 Encryption	423
24.5 Configuration with YaST	423
24.6 Tips and Tricks for Setting Up a WLAN	432
24.7 Troubleshooting	433

24.8 For More Information	435
---------------------------------	-----

25 Using NetworkManager 437

25.1 Use Cases for NetworkManager	437
25.2 Enabling or Disabling NetworkManager	438
25.3 Configuring Network Connections	439
25.4 Using the KDE NetworkManager Front-End	442
25.5 Using GNOME NetworkManager	446
25.6 NetworkManager and VPN	449
25.7 NetworkManager and Security	450
25.8 Frequently Asked Questions	451
25.9 Troubleshooting	453
25.10 For More Information	454

26 Using Tablet PCs 457

26.1 Installing Tablet PC Packages	458
26.2 Configuring Your Tablet Device	458
26.3 Using the Virtual Keyboard	459
26.4 Rotating Your Display	459
26.5 Using Gesture Recognition	460
26.6 Taking Notes and Sketching with the Pen	462
26.7 Troubleshooting	464
26.8 For More Information	465

27 Copying and Sharing Files 467

27.1 Scenarios	468
27.2 Access Methods	469
27.3 Accessing Files Using a Direct Connection	470
27.4 Accessing Files on Different OS on the Same Computer	471

27.5 Copying Files between Linux Computers	473
27.6 Copying Files between Linux and Windows Computers with SSH	480
27.7 Sharing Files between Linux Computers	481
27.8 Sharing Files between Linux and Windows with Samba	484
27.9 For More Information	486

A An Example Network 489

B Licences GNU 491

B.1 GNU General Public License	491
B.2 GNU Free Documentation License	494

À propos de ce guide

Ce manuel vous permet d'acquérir une compréhension générale de openSUSE®. Il est principalement destiné aux administrateurs système et aux particuliers avec une connaissance basique de l'administration système. Regardez les différentes parties de ce manuel pour une sélection d'applications nécessaires dans la vie de tous les jours et des descriptions détaillées de scénarios avancés d'installation et de configuration.

Scénarios de déploiements avancés

Apprenez comment déployer openSUSE à distance et soyez au courant des scénarios complexes de paramétrage de disques.

Gestion et mise à jour des logiciels

Comprendre comment installer ou supprimer des logiciels soit avec YaST, soit en utilisant la ligne de commande, comment utiliser le 1-Click Install et comment conserver votre système à jour.

Administration

Apprenez comment configurer et mettre à niveau openSUSE, comment administrer votre système en mode texte et quelques utilitaires importants pour les administrateurs Linux.

Système

Obtenez une introduction aux composants de votre système Linux et une compréhension plus poussée de leurs interactions.

Services

Apprenez comment configurer les divers services réseau et fichiers fournis par openSUSE.

Mobilité

Obtenez une introduction à l'informatique mobile avec openSUSE, apprenez les divers options pour l'informatique sans fil et la gestion d'énergie.

Plusieurs chapitres de ce manuel contiennent des ressources de documentation supplémentaires. Cela inclut la documentation disponible sur le système ainsi que 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 (↑Start-Up)

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 (p. i)

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 docu-

mentation 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 et Déploiement

Installation avec YaST

Installez votre système avec `yast`, outil central pour l'installation et la configuration de votre syst vous guide analyse affiche une vue d'ensemble de toutes les Help pour voir ces textes.

Si vous êtes débutant avec openSUSE, vous souhaiterez, peut-être, suivre les propositions de YaST par défaut pour la plupart de ces étapes, mais vous pouvez aussi ajustez les réglages comme décrit dans cette section afin d'ajuster votre système au plus près de vos préférences. Beaucoup de configurations basiques du système, telles que la création de compte utilisateur ou la langue du système, pourront être modifiées après le processus d'installation.

1.1 Choix du Média d'installation

Lors de l'installation d'openSUSE, choisissez depuis parmi les différents média disponible en ligne ou dans la version boîte.

Détails du DVD

Dans chaque boîtes DVD contenant la distribution openSUSE pour systèmes 32 et 64 bits. Le second disque contient les add-ons propriétaire.

Cette option d'installation ne requiert aucun accès réseau pour l'installation, ni ne nécessite la mise en place d'un dépôt externe pour l'installation complète d'openSUSE. Vous pouvez, cependant, rendre le contenu du DVD disponible sur un serveur d'installation et à travers de votre réseau.

Téléchargement du DVD

Un DVD5, est disponible par téléchargement pour les systèmes 32 et 64 bits.

Choisir cette option d'installation si vous voulez un système complet openSUSE. Mise à part pour le téléchargement, il n'est besoin d'aucune connexion réseau pour réaliser l'installation. Une fois que l'image ait été complètement téléchargée et que son support ait été créé, vous pourrez commencer l'installation. Vous pouvez rendre le contenu des DVD disponible sur un serveur d'installation et distribuée à travers votre réseau.

Le Live-CD KDE4/GNOME

Cette version Live-CD, disponible en téléchargement, inclue les gestionnaires de bureau GNOME ou KDE avec leur applications les plus populaires pour système 32 et 64 bits.

Choisir cette option d'installation pour expérimenter openSUSE. La version live-CD s'exécute sur votre ordinateur en se chargeant en RAM sans porter atteinte à votre disque dur et sans aucune installation. Vous pouvez par contre, installer openSUSE depuis le live-CD. Il n'est besoin d'aucune connexion réseau une fois que le live-cd est téléchargé pour faire cette installation.

ASTUCE : Démarrage du live CD depuis une clef USB.

L'image ISO du live CD peut-être aussi installé sur une clef USB bootable. Celle-ci est créée en utilisant le programme en ligne de commande `dd` avec la syntaxe suivante :

```
dd if=ISO_IMAGE of=USB_STICK_DEVICE bs=4M
```

`dd` is available on Linux and MacOS by default. A Microsoft Windows* version can be downloaded from <http://www.chrysocome.net/dd>.

!! Attention : L'utilisation de la commande `dd` effacera toutes les données sur votre périphérique USB !!

Mini-CD

Le mini-CD contient la version minimal de linux nécessaire à l'exécutif de l'installation. Le système d'installation et les données seront téléchargées depuis une source réseau. Pour installer depuis un réseau fournissant SLP, démarrez l'installation de la manière décrite dans la Section 1.2.1, « Installation depuis un serveur réseau utilisant SLP » (p. 7). Pour installer depuis un serveur HTTP,

FTP, NFS ou SMB suivez les instructions dans la Section 1.2.2, « Installation depuis un source réseau sans SLP » (p. 8).

IMPORTANT : Add-On du CD d'installation des logiciels additionnels

Bien que les CD de add-on (extensions ou produits tiers) ne peuvent pas être utilisés comme support d'installation autonome, ils peuvent être intégrés en tant que sources de logiciels supplémentaires pendant l'installation. Actuellement des CD avec la prise en charge de langages supplémentaires et des logiciels non open-sources sont disponibles en tant que CD de Add-on pour openSUSE. Référez-vous à la Section 1.7.1, « Produits Additionnels » (p. 15) pour de plus amples informations.

1.2 Choisir la méthode d'installation

Après avoir choisi le type de média d'installation, déterminez la méthode d'installation appropriée et les options de démarrage qui correspondent le mieux à ce dont vous avez besoin.

Installation depuis le média openSUSE

Choisissez cette option si vous voulez effectuer une installation autonome et qui n'ait besoin d'aucune connexion avec un réseau pour lui fournir les données d'installation ou l'infrastructure de démarrage. L'installation se fera comme décrit dans la Section 1.3, « L'Automatisation des Processus (Workflow) d'Installation » (p. 9).

Installation depuis le live-CD

Pour installer depuis le live-cd démarré le système depuis celui-ci. Depuis le live système, lancer la routine d'installation en cliquant sur l'icône *Install* du bureau. La première phase de l'installation se déroulera depuis une fenêtre sur le bureau. Il n'est pas possible de mettre à niveau un système existant avec un live CD, vous pouvez seulement effectuer une nouvelle installation (avec une configuration automatique).

Installation depuis un serveur réseau

Choisissez cette option si vous avez un serveur d'installation disponible sur votre réseau ou si vous souhaitez utiliser un serveur externe comme source

d'installation. Cette configuration peut-être paramétrée pour démarrer depuis un support physique (disquette, CD/DVD ou Disque Dur) ou paramétré pour démarrer en utilisant PXE/BOOTP. Référez-vous à la Section 1.2.1, « Installation depuis un serveur réseau utilisant SLP » (p. 7), Section 1.2.2, « Installation depuis un source réseau sans SLP » (p. 8), ou Chapitre 2, *Installation à distance* (p. 37) pour des détails.

Installing with openSUSE 12.3 Installer from Windows

Choose this installation option if you prefer a smooth transition from using Windows to using Linux. openSUSE 12.3 Installer allows you to boot into the openSUSE installation right from a running Windows by modifying the Windows boot loader. This installation option is only available from the DVD media. Refer to Section 1.2.3, « Installing with the openSUSE 12.3 Installer from Windows » (p. 8) for details.

openSUSE supports several different boot options from which you can choose, depending on the hardware available and on the installation scenario you prefer. Booting from the openSUSE media or using openSUSE 12.3 Installer are the most straightforward options, but special requirements might call for special setups:

Tableau 1.1 Les Options de Démarrage

Option de Démarrage	Description
DVD	C'est l'option la plus facile. Cette option peut être utiliser si le système à un lecteur de DVD qui est supporté par Linux.
openSUSE 12.3 Installer	openSUSE 12.3 Installer est installé dans une fenêtre Microsoft Windows et rend possible le démarrage direct de l'installation depuis ce système.
PXE ou BOOTP	Booting over the network must be supported by the system's BIOS or firmware, and a boot server must be available in the network. This task can also be handled by another openSUSE system. See

Option de Démarrage	Description
	http://en.opensuse.org/SDB:PXE_boot_installation for more information. Refer to Chapitre 2, <i>Installation à distance</i> (p. 37) for more information.
Disque Dur	L'installation d'openSUSE peut aussi être réalisée depuis un disque dur. Pour ça, copiez le noyau (<code>linux</code>) et le système d'installation (<code>initrd</code>) depuis le répertoire <code>/boot/architecture/</code> du média d'installation dans le disque dur et ajoutez l'entrée dans le boot-loader existant de l'installation openSUSE existante.

ASTUCE : Démarrage depuis le DVD sur une machine UEFI

►amd64 **em64t**: Le DVD1 peut-être utilisé comme support de démarrage pour une machine équipée avec UEFI (Unified Extensible Firmware Interface). Référez-vous à la documentation du constructeur pour des informations plus spécifiques. Si le démarrage échoue, essayez d'activer CSM (Compatibility Support Module) depuis votre firmware. ◀

1.2.1 Installation depuis un serveur réseau utilisant SLP

Si la configuration de votre réseau supporte OpenSLP et votre source d'installation réseau a été configurée pour s'annoncer elle-même via SLP (de " la façon décrite dans Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46)), démarrez votre système, pressez F4 dans l'écran de démarrage et choisissez *SLP* depuis le menu.

Le programme d'installation configure la connexion réseau avec DHCP et récupère l'emplacement de la source d'installation réseau depuis le serveur OpenSLP. Si la

configuration Automatique par DHCP échoue, vous êtes invité à entrer les paramètres appropriés manuellement. L'installation se déroulera comme décrit ci-dessous à l'exception de phase de configuration réseau qui sera demandé prioritairement pour l'ajout d'un dépôt additionnel. Cette étape n'est pas nécessaire vu que le réseau est déjà censé être configuré et actif à ce stade.

1.2.2 Installation depuis un source réseau sans SLP

Si la configuration de votre réseau ne supporte pas OpenSLP pour la récupération des sources, démarrer le système et pressez F4 à l'écran de démarrage pour choisir le protocole réseau souhaité (NFS, HTTP, FTP ou SMB/CIFS). Fournissez l'adresse du serveur et le chemin d'installation vers le média d'installtion.

Le programme d'installation configure automatiquement la connexion réseau avec DHCP. Si cette configuration échoue, vous serez invité a entrer les paramètres appropriés manuellement. L'installation récupère les données dont elle a besoin depuis la source spécifiée. L'installation procède ensuite comme décrit plus bas à l'exception de la configuration réseau nécessaire pour l'ajout dépôts supplémentaires. Cette étape n'est pas nécessaire vu que le réseau est déjà censé être configuré et actif à ce stade.

1.2.3 Installing with the openSUSE 12.3 Installer from Windows

openSUSE 12.3 Installer is a Microsoft Windows application that prepares your computer to directly boot into the openSUSE installation without having to adjust BIOS settings. It is only available on DVD media. To use the installer, insert the openSUSE media under Windows. The openSUSE 12.3 Installer setup automatically starts (if not, run `openSUSE11_2_LOCAL.exe` from the DVD). Choose a language for the installation and follow the instructions on the screen. The language you choose here is also preconfigured to be used for the openSUSE installation.

On the next reboot, the Microsoft Windows boot loader launches. Choose *openSUSE 12.3 Installer* to start the openSUSE installation. In order to proceed with the installation, you will be prompted to insert the installation media. The installation proceeds as described below. When Microsoft Windows is booted again, openSUSE 12.3 Installer is automatically uninstalled.

ASTUCE : Installation d'openSUSE aux côté de Microsoft Windows

openSUSE peut aisément être installé aux côtés de Microsoft Windows. Effectuer l'installation comme décrit ci-après, une installation pré-existante de Windows sera automatiquement détectée et une option dual boot sera installée. Si Windows est installé sur la totalité du disque sur lequel se fait l'installation, la routine d'installation vous prosera de réduire la partition sur laquelle Windows est installé afin de ménager une place pour installer openSUSE. Référez-vous à Section 1.10.1.1, « redimensionner une partition windows » (p. 20) avant de commencer pour des informations détaillées.

1.3 L'Automatisation des Processus (Workflow) d'Installation

L'installation d'openSUSE se déroule en trois phases : préparation, installation et configuration. Durant la phase de préparation vous configurez certains paramètres basiques tels que la langue, l'heure, type de bureau, utilisateurs, mots de passe , configuration des disques durs, et ampleur de l'installation. Pendant la phase non-interactive de l'installation le logiciel est installée et le système est préparé pour son premier démarrage. A la fin de l'installation la machine redémarre sur son nouveau système et commence la configuration finale de celui-ci. Vous pouvez choisir entre une configuration totalement automatisée ou manuelle. A ce stade, le réseau et l'accès internet, de même que les composants matériels, sont configurés.

1.4 Démarrage du système pour l'installation

Vous pouvez installer openSUSE depuis des sources d'installation locales, tels que les CD et DVD d'openSUSE ou depuis la source réseau d'un serveur FTP, HTTP, NFS ou SMB. Toutes ces approches requierent un accès physique au système à installer ainsi qu'une interaction de l'utilisateur. La procédure d'installation est basiquement la même mise à part pour la source d'installation. Toutes les exceptions sont suffisamment mis en lumière dans la description de workflow suivante.

1.5 l'Ecran de Démarrage

The boot screen displays a number of options for the installation procedure. *Boot from Hard Disk* boots the installed system and is selected by default, because the CD is often left in the drive. Select one of the other options with the arrow keys and press Entrée to boot it. The relevant options are:

Installation

Le mode d'installation normale. Toutes les fonctions modernes sont activées.

Au cas où l'installation venait à échouer, voir « F5 Kernel » (p. 12) pour les options de démarrage qui désactive des fonctions éventuellement problématiques.

Système de Secours

Démarre un système Linux minimal sans interface graphique utilisateur. Pour plus d'informations, voir section « Using the Rescue System » (Annexe A, *Aide et problèmes*, ↑Start-Up). Cette option n'est pas disponible sur les live-CD.

Vérifier le Média d'Installation

Cette option n'est seulement disponible quand le support d'installation est créé à partir d'une image ISO téléchargée. Dans ce cas il est recommandé de vérifier l'intégrité du fichier ISO d'installation. Cette option démarre le système d'installation en vérifiant auparavant le fichier ISO. Dans le cas où la vérification se fait avec succès, l'installation proprement dites commence. Si une corruption est détectée, l'installation s'annule.

Test du Firmware

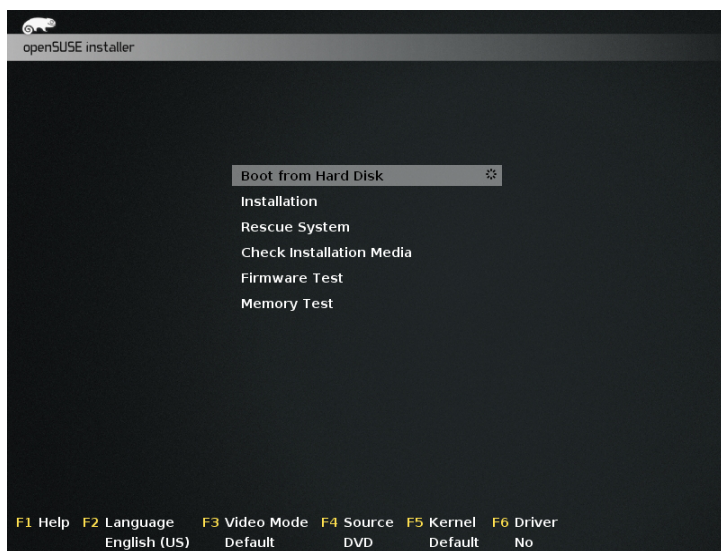
Démarrez le vérificateur du BIOS qui validera l'ACPI et les autres composantes de votre BIOS. Cette option n'est pas disponible sur les live-CD.

Test de la Mémoire

Le test de votre RAM se fait en répétant des cycles de lecture et écriture.

Terminer le test en redémarrant. Pour plus d'informations, voir section « Fails to Boot » (Annexe A, *Aide et problèmes*, ↑Start-Up). Cette option n'est pas disponible sur les live-CD.

Figure 1.1 *l'Ecran de Démarrage*



Utilisez les touches de fonctions indiquées dans la barre en bas de l'écran pour changer la langue, la résolution de l'écran, la source d'installation ou pour ajouter un lecteur additionnel depuis votre fournisseur de matériel:

F1 *Help*

Get context-sensitive help for the active element of the boot screen. Use the arrow keys to navigate, Entrée to follow a link, and Esc to leave the help screen.

F2 *Language*

Choisit la langue d'affichage et la disposition clavier correspondante pour l'installation. La langue par défaut est l'anglais (US).

F3 *Video Mode*

Choisit entre plusieurs modes d'affichage graphique. Choisir *Text Mode* si l'installation graphique cause des problèmes.

F4 *Source*

Normalement, l'installation s'effectue depuis le support d'installation inséré. Ici, vous pouvez choisir d'autres sources, tcomme un serveur FTP ou NFS. Si l'installation est déployée sur une réseau accueillant un serveur SLP, choisissez une source d'installation disponible sur le serveur avec cette option.

Vous trouverez plus d'information dans le Chapitre 14, *SLP Services in the Network* (p. 279).

F5 *Kernel*

Si vous rencontrez des problèmes avec l'installation régulière; ce menu offre la possibilité de désactiver quelques fonctions potentiellement problématiques. Si votre matériel ne supporte pas ACPI (advanced configuration and power interface: Gestion avancée de l'alimentation); choisissez *No ACPI* pour une installation sans le support de l'ACPI. *No local APIC* Désactive le support de APIC (Advanced Programmable Interrupt Controllers : Contrôleur programmable avancé d'interruption programmable) qui peut causer des dysfonctionnement avec certains matériels. *Safe Settings* Démarre le système sans le mode DMA (pour les lecteurs CD et DVD) et sans la gestion de l'alimentation.

Si vous n'êtes pas sûr, essayez les options suivantes en premier : *Installation—ACPI Disabled* or *Installation—Safe Settings*. Les experts peuvent aussi utiliser la ligne de commande (*Boot Options*) pour entrer ou changer des paramètres du noyau.

F6 *Driver*

Pressez cette touche pour notifier au système une mise à jour optionnelle de pilote pour openSUSE. Depuis *File* ou *URL*, chargez les pilotes directement avant que l'installation ne démarre. Si vous choisissez *Yes*, vous êtes invité à insérer le disque de mise à jour à l'étape appropriée dans le processus d'installation.

F7 *Arch*

Si vous installez depuis un support d'installation supportant les architecture 32 et 64 bits et possédez un processeur supportant le 64 bits, choisissez si vous souhaitez installer un système 64 bits ou 32 bits. Par défaut, un système 64 bits sera installé sur un computer 64 bit. Pour installer un système 32 bits, pressez F7 et choisissez `guimenu>32bit`

ASTUCE : Utiliser IPv6 pendant l'installation

Par défaut vous ne pouvez seulement assigné des adresse IPv4 à votre machine. Pour activer IPv6 pendant k'installation, entrez un des paramètres suivants à l'invite de commande : `ipv6=1` (IPv4 et IPv6) ou `ipv6only=1` (IPv6 seulement).

Après le démarrage de l'installation, openSUSE charge et configure un système Linux minimal pour exécuter la procédure d'installation. Pour voir les message de démar-

rage et les notices de copyright pendant ce processus, pressez **Esc**. A la fin de ce processus, le programme d'installation de YaST démarre et affiche l'écran graphique de l'installateur.

ASTUCE : Installation sans souris

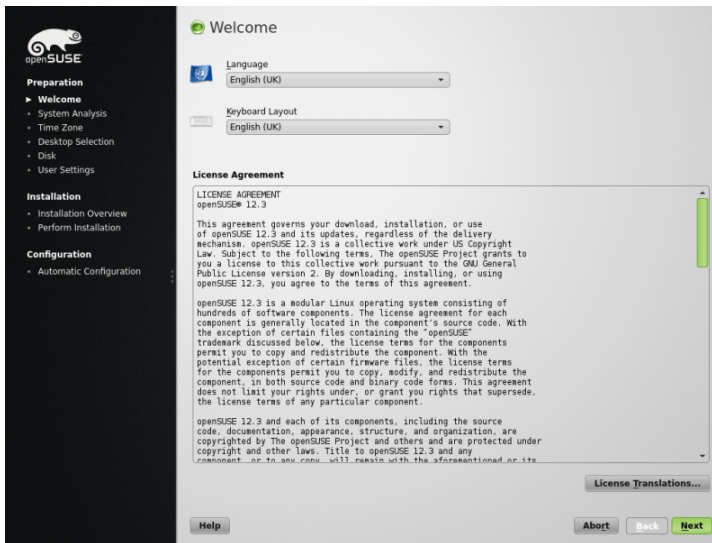
Si l'installateur ne détecte pas votre souris correctement, utilisez **Tab** pour naviguer, les touches flèches pour faire défiler et **Enter** pour confirmer une sélection. Plusieurs boutons ou champs de sélection contiennent une lettre et un caractère de soulignement (underscore). Utilisez **Alt + Letter** pour choisir un ceux-ci directement plutôt que naviguer vers eux en utilisant **Tab**.

1.6 Bienvenue

Démarrez l'installation d'openSUSE en choisissant votre langage. Changer la langue préselectionnera automatiquement une disposition clavier correspondante. Vous pouvez changer cette proposition par défaut en sélectionnant une autre disposition clavier depuis le menu déroulant. La langue choisie ici est aussi utilisé pour adopter une zone horaire pour l'horloge système. Ces réglages — ainsi que la sélection des langues secondaires à installer sur votre système— peuvent être modifiés plus-tard dans le *Résumé de l'Installation*, décrit à la Section 1.12, « Paramètres d'Installation » (p. 26). Pour plus d'information sur les réglages de la langue dans le système installé, voir Chapitre 11, *Changing Language and Country Settings with YaST* (↑Start-Up).

Lisez soigneusement le contrat de licence qui s'affiche sous la sélection de la langue et de la disposition clavier. Utilisez *License Translations...* pour accéder aux traductions. Si vous êtes d'accord avec les termes du contrat, cliquez *Next* pour procéder à l'installation. Si vous n'êtes pas d'accord avec le contrat de licence cliquez sur *Abort* pour terminer l'installation.

Figure 1.2 *Bienvenue*



1.7 Mode installation

Après l'analyse du système (Pendant laquelle YaST sonde les périphériques de stockage et essaie de trouver les autres systèmes déjà installés sur votre machine) les modes d'installation disponibles sont affichés. Cette étape est sautée quand l'installation se fait depuis un live-CD, puisque celle-ci qu'une nouvelle installation avec configuration automatique.

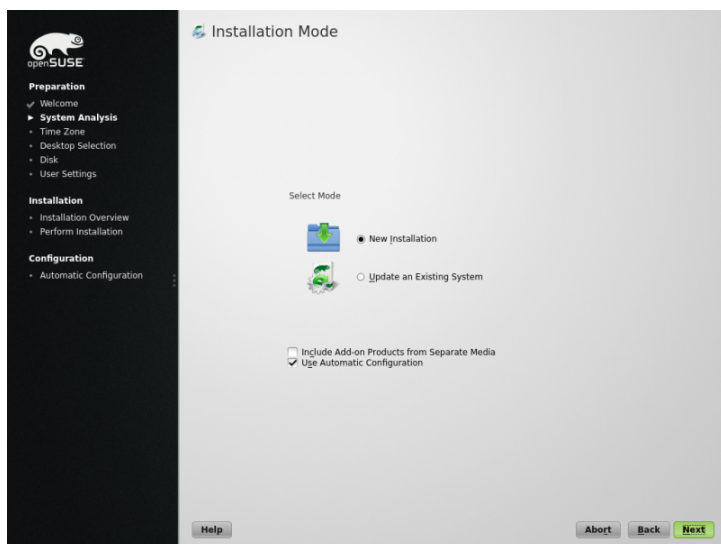
Nouvelle installation

Choisissez cette option pour démarrer une nouvelle installtion à partir de zéro.

Mise à niveau

Select this option to update an existing installation to a newer version. For more information about system updates, see Chapitre 16, *Upgrading the System and System Changes* (↑Start-Up).

Figure 1.3 *Mode installation*



La configuration automatique est choisie par défaut lors de l'exécution d'une nouvelle installation. Dans ce mode le système configure automatiquement votre matériel et le réseau, aussi celle-ci demande un minimum d'interaction. Si nécessaire, vous pouvez changer toute configuration qui sera paramétré par la suite dans le système installé en utilisant YaST décochez *Use Automatic Configuration* si vous préférez faire une configuration manuelle durant l'installation.

Check *Include Add-On Products from Separate Media* to include add-on products during the installation. An add-on product can include extensions, third-party products and drivers or additional software for your system such as support for additional languages.

Cliquez *Next* pour continuer. Si vous avez choisi d'inclure un produit additionnel, continuer avec Section 1.7.1, « Produits Additionnels » (p. 15), sinon passez la prochaine section et rendez-vous à la Section 1.8, « Heure et fuseaux horaires. » (p. 17).

1.7.1 Produits Additionnels

Les add-ons peuvent être installés soit depuis une source local (CD, DVD ou un répertoire), soit depuis une source réseau (HTTP, FTP, NFS, CIFS,...). Lors de

l'installation depuis un source réseau, vous aurez besoin de premièrement configurer le réseau (sauf si vous effectuez une installation réseau dans ce cas le réseau déjà configuré sera utilisé). Choisissez *Yes*, *Run the Network Setup* et procédez de la façon décrite dans Section 1.7.1.1, « Configuration Réseau » (p. 16). Si le produit additionnel est disponible localement, choisissez *No*, *Skip the Network Setup*.

Cliquez *Next* et spécifiez la source du produit. Les types de sources disponibles sont *CD*, *DVD*, *Disque Dur*, *Stockage de Masse USB*, un *Répertoire local* ou une *image ISO locale* (Si aucun réseau n'est configuré). Si le produit additionnel est disponible sur un disque dur, choisir *Disque Dur* pour installer depuis un disque dur qui n'est pas monté, ou *Repertoire Local/Image ISO Locale* pour installer depuis le système de fichier local. Les add-ons peuvent être distribué depuis un dépôt ou un jeu de fichiers rpm. Dans ce dernier cas, cochez *Plain RPM Directory*. Chaque fois qu'un réseau est disponible, vous pouvez choisir depuis des sources additionnelles distantes telle que HTTP, SLP, FTP, etc... Il est aussi possible de spécifier une URL directement.

Cochez *Download Repository Description Files* pour télécharger les fichiers décrivant les dépôts maintenant. Si cette option n'est pas choisie, ils seront télécharger au moment du démarrage de l'installation. Continuez avec *Next* et insérez un CD ou un DVD si cela est requis. Selon le contenu du add-on il sera peut être nécessaire d'accepter des accords de licence supplémentaires.

Il est aussi possible de configurer les produits additionnels plus tard. L'utilisation des add-ons sur un système déjà installé est décrite dans Chapitre 8, *Installing Add-On Products* (↑Start-Up).

1.7.1.1 Configuration Réseau

Au moment de l'invocation de la configuration réseau YaST recherche les cartes réseau disponibles. Si plus d'une carte sont découvertes, vous devez choisir la carte à configurer depuis une liste.

Si un adaptateur réseau ethernet n'est pas déjà connecté, un avertissement apparaîtra. Assurez vous que le câble réseau est branché sur celui-ci et choisissez *Yes*, *Use It*. Si votre réseau est équipé d'un serveur DHCP, choisir *Configuration automatique (via DHCP)*. Pour paramétrer automatiquement l'adresse réseau choisir *Configuration statique* et spécifier *Adresse IP*, *Masque de sous-réseau*, *Passerelle par défaut*, et *l'adresse de serveur DNS*.

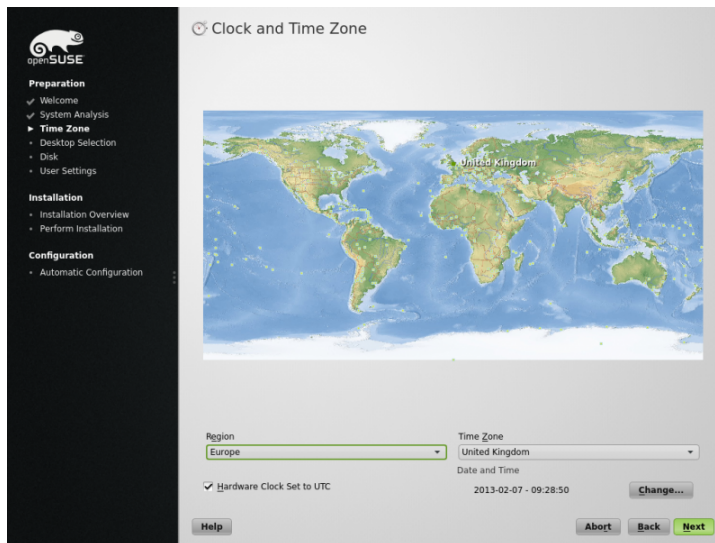
Certains réseau requiert l'utilisation d'un serveur proxy pour accéder à internet. cocher la case *Utiliser un Proxy pour accéder à internet* et entrez les paramètres appro-

priés. Cliquez sur *Acceptez* pour procéder à la configuration du réseau. La procédure d'installation continuera avec l'installation des produits additionnels ou la configuration d'un dépôt comme décrit à la Section 1.7.1, « Produits Additionnels » (p. 15).

1.8 Heure et fuseaux horaires.

Dans cette fenêtre, choisissez votre zone géographique et votre fuseau horaire. Les deux sont préselectionnés selon la langue d'installation. Pour changer les valeurs préselectionnées, utiliser soit la carte ou le menu déroulant du champ *Region* et *Fuseau horaire*. Pour utiliser la carte, placer le curseur vaguement dans la direction de votre zone géographique et faite un clic gauche pour zoomer. Puis choisir votre pays ou région en cliquant à gauche. Clic droit pour retourner à la carte globale.

Figure 1.4 *Heure et fuseaux horaires.*



Pour paramétrer l'heure, choisir soit *Horloge système réglée sur l'UTC*. Si vous exécutez un autre système d'exploitation sur votre machine, tel qu'un Microsoft Windows, il est probable que votre système utilise plutôt l'heure locale. Si vous exécutez seulement un Linux sur votre machine, paramétrez l'heure en UTC et choisissez si le passage de l'heure normale à l'heure d'été doit se faire automatiquement.

Si un réseau est déjà configuré, l'heure est automatiquement paramétrée via un serveur NTP (Network Time Protocol). Cliquez *Changer* pour modifier les para-

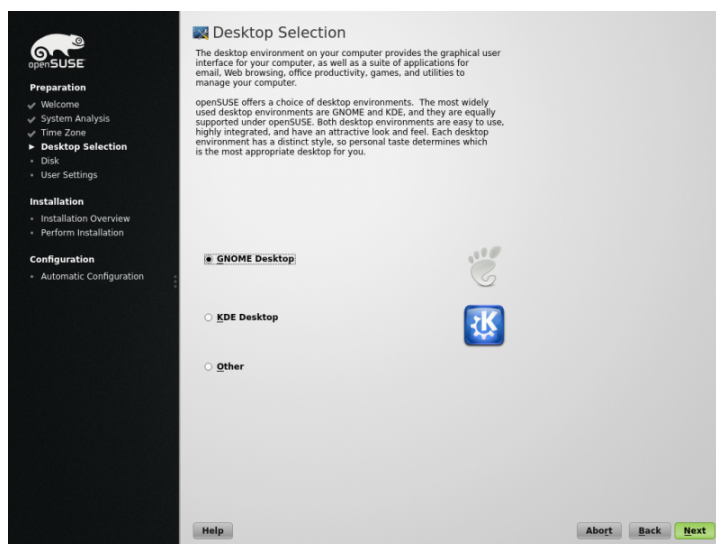
mètre NTP, ou pour *Manuellement* régler l'heure. Consultez le Chapitre 17, *Time Synchronization with NTP* (p. 317) pour plus d'informations sur la configuration des services NTP. Quand vous avez fini, cliquez sur *Acceptez* pour continuer l'installation.

1.9 Sélection du bureau

Avec openSUSE, vous pouvez choisir parmi plusieurs gestionnaires de bureau. Le principaux, *KDE* et *GNOME*, sont des bureaux graphiques puissants similaires à windows. Cette étape est sautée quand vous installez depuis un live-CD, puisque ce type d'installation est déjà préconfigurée pour utiliser soit KDE ou GNOME.

Si vous préférez un autre gestionnaire de bureau différent, choisissez *Autres* pour plus d'options. Les bureaux *XFCE* et *LXDE* sont des environnement rapide et léger adaptés au machines les plus modestes. Avec *Minimal X Window*, installez un gestionnaire de fenêtrage graphique, permettant de faire tourner des applications X autonomes et des fenêtres de console mais ne fournissant pas les fonctionnalités usuelles d'un environnement intégré. Dans *Minimal Server Selection (Text Mode)*, seule la console terminal est disponible.

Figure 1.5 *Sélection du bureau*



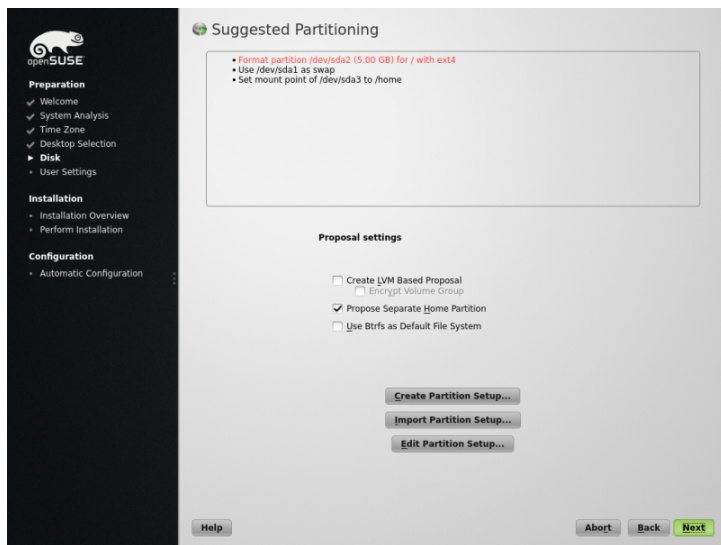
1.10 Partitionnement suggéré

Définissez une partition pour openSUSE à cette étape. Dans la plupart des cas, un système de partition raisonnablement acceptable sans modification est proposé. Si le disque dur sélectionné comme cible de l'installation ne contient que des partitions NTFS ou FAT, YaST propose de réduire une de ces partitions. Acceptez la proposition avec *Suivant* et procédez à l'installation. Les utilisateurs expérimentés peuvent aussi personnaliser la proposition ou effectuez leur propre partitionnement.

Le partitionnement proposé est *Basé Partition* par défaut. Si vous préférez une configuration *basée sur LVM*, choisir les options respectives pour automatiquement convertir la proposition. Référez vous à la Section 3.2, « Configuration LVM » (p. 86) pour plus d'informations sur le gestionnaire de volume logique LVM.

Pour faire de léger changement dans la configuration proposée (changer le système de fichier ou encrypter les partitions par exemple), choisissez *Editez la configuration des Partitions*. Voir Section 3.1, « Utilisation du partitionneur YaST » (p. 75) pour des instructions à ce sujet.

Figure 1.6 Partitionnement suggéré



1.10.1 Partitionner a disque spécifique

Si votre machine contient mplus d'un disque dur et que vous voulez restreindre le partitionnement proposé à un disque seulement, choisissez *Créez une configuration de Partition* puis sélectionnez un disque spécifique depuis la liste. Si le disque dur choisi ne contient pas encore de partition, l'utilisatio du disque entier sera suggérée. Autrement vous pouvez choisir quelle partition existante utiliser. Pour ajouter une partition séparée pour vos données personnelles choisissez *Proposez une partition home séparée*. Au lieu de la proposition par défaut basée sur des partitions, il est possible de *Créez une proposition basée sur LVM*. Cliquez deux fois sur *Suivant* pour continuer vers l'étape suivante.

1.10.1.1 redimensionner une partition windows

Si le disque dur sélectionné contient seulement une partition FAT ou NTFS Windows, YaST offre la possibilité d'effacer ou réduire cette partition. Si vous choisissez *Effacer Windows complètement*, la partition est marquée pour l'effacement et l'espace ainsi créé est utilisé pour l'installation d' openSUSE.

AVERTISSEMENT : Effacement de Windows

Si vous effecez Windows, toutes les données seront perdues à moins d'une sauvegarde avant le démarrage de l'installation.

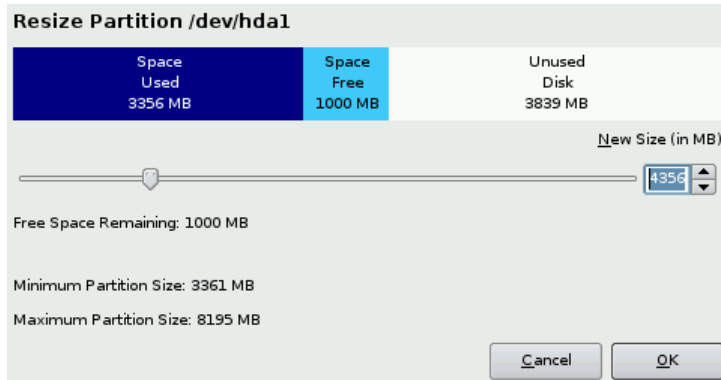
Pour *Réduire la partition Windows*, vous avez besoin d'interrompre l'installation et de démarrer Windows pour préparer la partition avant que celle-ci soit réduite. Pour tout système de fichiers Windows procédez comme suit :

1. Désactiver le fichier de `mémoire virtuelle`, s'il existe.
2. exécutez `scandisk`.
3. exécutez `defrag`.

Après ces préparations, redémarrez l'installation d'openSUSE. Quand vous êtes sur la configuration du partitionnement procédez comme précédemment et choisissez *Réduire la partition Windows* . Après une vérification rapide de la partition, le dialogue pour le redimensionnement de partitions windows s'ouvre.

La barre graphique montre combien d'espace disque est occupé par Windows et combien il en reste de disponible. Pour changer les réglages proposés utilisez le curseur ou le champs d'entrée pour ajuster la taille de la partition.

Figure 1.7 Redimensionnement de la partition Windows



Si vous quittez ce dialogue enchoisissant *Suivant>/guimenu>*, les réglages sont conservés et vous êtes renvoyer au dialogue précédent. Le redimensionnement réel aura lieu plus tard, avant le formatage du disque dur.

IMPORTANT : Ecriture sur une partition NTFS

By default, the Windows uses the NTFS file system. openSUSE includes read and write access to the NTFS file system, but this feature has a few limitations. This means that you cannot read or write encrypted or compressed files. Furthermore, the Windows file permissions are not honored at the moment. See <http://en.opensuse.org/SDB:NTFS> for more information.

1.10.2 Partitionnement personnalisé.

Set up your own partitioning scheme by selecting *Create Partition Setup* and then *Custom Partitioning*. The Expert Partitioner opens, displaying the current partition setup. Expand the *Hard Disks* in the System View pane by clicking on the +, then click on one of the listed hard disks. Now you can *Add*, *Edit*, *Resize*, or *Delete* partitions. For more information about custom partitioning and configuring advanced features, refer to Section 3.1, « Utilisation du partitionneur YaST » (p. 75)

1.11 Créez un nouvel utilisateur

Créez un utilisateur local à cette étape. L'administration d'utilisateurs locaux est une option appropriée pour les stations de travail autonomes. Pour la mise en place d'une authentification centralisée pour des clients réseau, cliquez *Changer* et continuer à la Section 1.11.1, « Réglages Experts » (p. 24).

Après avoir entré le nom et le prénom, soit vous acceptez la suggestion soit vous spécifiez un nouveau *nom d'utilisateur* de login. Enfin, entrez un mot de passe pour l'utilisateur. Réentrez celui-ci pour confirmer (afin de vous assurer que vous n'avez pas fait d'erreur). Pour fournir une sécurité efficace, un mot de passe doit comporter cinq à huit caractères. La taille maximale pour un mot de passe est de 72 caractères. Cependant, si aucun module spécial de sécurité n'est chargé, seules les huit premiers caractères sont utilisés pour discerner le mot de passe. Les mots de passe sont sensibles à la casse. Les caractères spéciaux (ASCII sur 7 bits) et les chiffres de 0 à 9 sont autorisés. Les autres caractères spéciaux tels que les trémas ou caractères accentués ne sont pas autorisés.

La fiabilité des mots de passe saisis sont vérifiés. Quand vous entrez un mot de passe facile à deviner (un mot de dictionnaire ou un nom) vous voyez apparaître un avertissement. C'est une bonne pratique de sécurité de fournir des mots de passe forts.

IMPORTANT : Nom d'utilisateur et mot de passe

Rappelez-vous du nom d'utilisateurs et du mot de passe associé car il vous seront demandés chaque fois que vous vous connecterez au système.

Figure 1.8 *Créez un nouvel utilisateur*

openSUSE

Preparation

- ✓ Welcome
- ✓ System Analysis
- ✓ Time Zone
- ✓ Desktop Selection
- ✓ Disk
- ▶ **User Settings**

Installation

- Installation Overview
- Perform Installation

Configuration

- Automatic Configuration

Create New User

User's Full Name

Username

Password

Confirm Password

☒ Use this password for system administrator
☐ Receive System Mail
☒ Automatic Login

Summary

The authentication method is local /etc/passwd.
The password encryption method is SHA-512.

[Change...](#)

[Help](#) [Abort](#) [Back](#) [Next](#)

Trois options supplémentaires sont disponibles :

Utiliser ce mot de passe pour l'administration du système

Si cette option est choisie, le même mot de passe sera utilisé pour l'administrateur du système `root` et l'utilisateur que vous venez de créer. Cette option est adaptée pour les stations de travail autonomes ou pour les machines faisant parti d'un réseau domestique administrées par un seul utilisateur. Quand l'option n'est pas cochée vous êtes invité à saisir un mot de passe administrateur à la prochaine étape de l'installation. (voir Section 1.11.2, « Mot de passe pour l'administrateur système `root` » (p. 25)).

Recevoir des Mail du Système

En cochant cette case des messages créés par le système sont envoyés par le système aux utilisateurs. Ceux-ci sont habituellement envoyés seulement au `root`, l'administrateur du système. Cette option est utile au compte les plus fréquemment utilisés, car il est fortement recommandé de se connecter en `root` seulement lors d'opérations exceptionnelles.

Les mails envoyés par le système sont stockés dans la boîte mail locale `/var/spool/mail/nom_utilisateur`, où `nom_utilisateur` est le nom de login de l'utilisateur choisi. Pour lire les mails après l'installation, vous pouvez utiliser un client de messagerie, Kmail ou Evolution par exemple.

Automatic Login

Cette option connecte automatiquement l'utilisateur courant au système quand celui-ci démarre. Ceci est principalement utile si la machine est utilisée par un seul utilisateur.

AVERTISSEMENT : Automatic Login

Avec le login automatique activé, le système démarre directement sur votre bureau sans aucune authentification. Si vous stockez des informations sensibles sur votre système, vous ne devez pas activer cette option tant que votre ordinateur peut être aussi utilisé par d'autres personnes.

1.11.1 Réglages Experts

Cliquez *Changer* dans le dialogue de création d'utilisateurs pour paramétrer une authentification réseau ou, si besoin, importer des utilisateurs depuis une installation précédente. Vous pouvez aussi changer le type de cryptage du mot de passe dans ce dialogue.

Vous pouvez aussi ajouter des utilisateurs supplémentaires ou changer la méthode d'authentification des utilisateurs dans le système installé. pour des informations détaillées sur la gestion des utilisateurs, voir Chapitre 10, *Managing Users with YaST* (↑Start-Up).

La méthode d'authentification par défaut est *Local (/etc/passwd)*. Si une ancienne version d'openSUSE ou un autre système utilisant */etc/passwd* est détectée, vous pouvez importer les utilisateurs locaux. Pour ce faire, choisir *Lire les données utilisateur depuis une installation précédente* et cliquez *Choisir*. Dans le prochain dialogue, sélectionnez les utilisateurs à importer et terminer avec *OK*.

L'accès aux services d'authentification réseau suivants peuvent être configuré :

LDAP

Utilisateurs sont administré centralement grâce à un serveur LDAP pour toutes les machines sur le réseau. Plus d'informations sont disponibles dans section « Configuring an LDAP Client with YaST » (Chapitre 4, *LDAP—A Directory Service*, ↑Security Guide).

NIS

Les utilisateurs sont administrés centralement par un serveur NIS pour toutes les machines sur le réseau. Voir section « Configuring NIS Clients » (Chapitre 3, *Using NIS*, ↑Security Guide) pour plus d'informations.

Domaine Windows

L'authentification SMB est souvent utilisée dans les réseau mixte Linus et Windows. et section « Configuring a Linux Client for Active Directory » (Chapitre 5, *Active Directory Support*, ↑Security Guide).

Avec l'administration des utilisateurs via *LDAP* et *NIS*, vous pouvez utiliser une authentification Kerberos. Pour cela, sélectionnez *Configurer l'Authentification Kerberos*. Pour plus d'informations sur Kerberos, référez-vous à Chapitre 6, *Network Authentication with Kerberos* (↑Security Guide).

1.11.2 Mot de passe pour l'administrateur système root

Si vous n'avez pas choisi *Utiliser ce mot de passe pour l'administrateur du système* dans l'étape suivante, vous serez invité à entrer un mot de passe pour l'administrateur système root user;. Sinon cette configuration sera sautée.

`root` est le nom du superutilisateur, ou l'administrateur du système. Contrairement aux utilisateurs réguliers (qui peuvent avoir ou ne pas avoir la permission d'exécuter certaines commandes ou accéder à certaines zones du système), `root` a un accès illimité pour changer la configuration du système, installer les programmes, et configurer le nouveau matériel. Si les utilisateurs oublient leur mot de passe ou ont d'autres problèmes avec le système, `root` peut les aider. Le compte `root` ne devrait être utilisé que pour l'administration du système, la maintenance, et les réparations. Se connecter quotidiennement avec un compte `root` est relativement risqué : une simple erreur peut conduire à des dommages irréversibles du système de fichier.

À des fins de vérification, le mot de passe du `root` doit être entré deux fois. Ne l'oubliez pas celui-ci. une fois entrée il ne peut être récupéré.

The `root` can be changed any time later in the installed system. To do so run YaST and start *Security and Users > User and Group Management*.

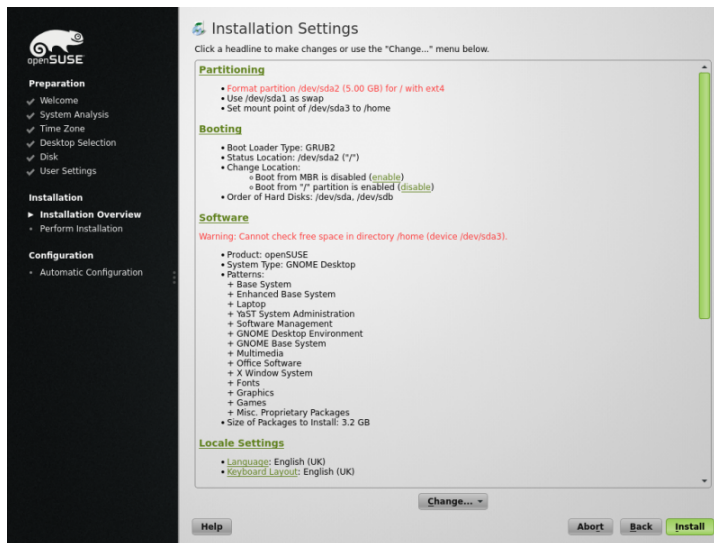
AVERTISSEMENT : L'utilisateur root

The `root` user has all the permissions needed to make changes to the system. To carry out such tasks, the `root` password is required. You cannot carry out any administrative tasks without this password.

1.12 Paramètres d'Installation

Durant la dernière étape de l'installation finale L'installation à proprement parlée commence, vous pouvez modifier les paramètres de l'installation suggéré par YaST ainsi que revoir les réglages que vous avez fait jusqu'ici. pour modifier les suggestions soit cliquez sur *Changer* et sélectionnez la catégorie pour changer, soit cliquez sur un des titres. Après la configuration d'un des objets présenté dans ces boîtes de dialogue, vous êtes toujours ramené à la fenêtre de réglaged des paramètres de l'installation, qui sera mise à jour conformément à vos réglages.

Figure 1.9 Paramètres d'Installation



ASTUCE : Restaurer les réglages pas défaut.

You can reset all changes to the defaults by clicking *Change > Reset to Defaults*. YaST then shows the original proposal again.

1.12.1 *Partitioning*

Revoyez et, si nécessaire, changez la configuration de la partition vous avez configuré précédemment. La modification de la partition permet soit de partitionner un disque spécifique, soit lorsque vous choisissez *Partition personnalisée*, appliquer votre propre schéma de partition.

1.12.2 *Booting*

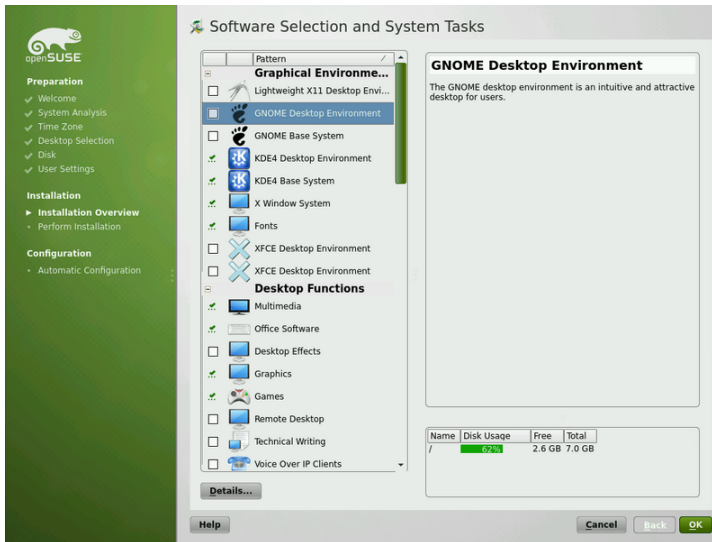
YaST propose une configuration de démarrage pour votre système. Les autres système d'exploitation trouvé sur votre machine, tel que des systèmes Microsoft ou Linux, seront automatiquement détectés et ajouté au boot loader. Cependant, openSUSE sera démarrée par défaut. Normalement, vous pouvez laisser ces réglages inchangés. Si vous avez besoin néanmoins besoin de personnaliser ces réglages, modifier la proposition pour votre système. Pour des informations, voir Section 9.2, « Configuring the Boot Loader with YaST » (p. 164). La méthode de démarrage doit être changé seulement par les utilisateurs expérimentés.

1.12.3 *Software*

openSUSE contient un certain nombre de modèles de pack logiciel pour des applications diverses. Cliquez *Software* pour commencer la sélection du modèle et modifiez le champ d'installation selon vos besoin. Sélectionner votre pattern dans la liste et voir leur description dans la fenêtre dans la partie droite de la fenêtre. Chaque modèle contient plusieurs packages logiciels nécessaire pour une fonction spécifique (e.g. Multimedia ou bureautique). Pour une description plus détaillée de la selection basée sur les packages à installer, sélectionnez *Details* pour passer au gestionnaire de logiciel YaST.

Vous pouvez aussi installer des packages logiciels additionnels ou retirer un package de votre système à tout moment avec le gestionnaire de logiciel YaST. Pour plus d'informations, référez-vous au Chapitre 5, *Installation et Désinstallation de Logiciels* (↑Start-Up).

Figure 1.10 Sélection des logiciels et des tâches du système



1.12.4 Locale Settings

Ici, vous pouvez changer la langue du système, ainsi que la *disposition du clavier* que vous avez défini à la première étape de l'installation. Il est aussi possible d'ajouter des langues additionnelles. Pour ajuster le langage du système, sélectionnez *Langage*. Choisissez un langage dans la liste. Le langage principal est utilisé comme langue du système. Vous pouvez aussi adapter la disposition du clavier et le fuseau horaire au premier langage si le réglage de base diffère. *Details* vous permet de modifier les réglages du langage pour l'utilisateur `root`, fournir le support de UTF-8, ou encore spécifier la langue (e.g sélectionner Anglais d'Afrique de Sud)

Choisir une langue secondaire pour avoir la possibilité de passer d'une langue à l'autre à tout moment sans avoir à installer de packages supplémentaires. Pour plus d'informations, voir Chapitre 11, *Changing Language and Country Settings with YaST* (↑Start-Up).

Pour changer la disposition du clavier, sélectionner *Keyboard Layout*. Par défaut, la disposition correspond au langage choisi pour l'installation. Sélectionnez la disposition clavier depuis la liste. Utiliser le champ *Test* dans le bas de la fenêtre de dialogue si vous entrez un caractère spécial le caractère correspondant doit s'afficher. Des options

pour le réglages fin sont disponibles dans *Mode Expert*. Quand vous avez fini, cliquez *Acceptez* pour retourner au résumé de l'installation.

1.12.5 *Time Zone*

Ajustez le fuseau horaire et le réglage de l'heure ici. A condition qu'un réseau soit configuré, vous pouvez aussi paramétrer un client NTP (Network Time Protocol) qui se synchronisera automatiquement avec un serveur de temps. C'est la même configuration dont nous avons parlé plus haut dans la Section 1.8, « Heure et fuseaux horaires. » (p. 17).

1.12.6 *Réglages utilisateur*

Changez les réglages *utilisateur* courant ou paramétrez le *mot de passe administrateur* ici. C'est la même configuration que celle évoquée plus tôt dans la Section 1.11, « Créez un nouvel utilisateur » (p. 22).

1.12.7 *Default Runlevel*

openSUSE can boot to different runlevels. Normally, there should be no need to change anything here, but if necessary set the default runlevel with this dialog.

1.12.8 *System*

Cette fenêtre présente toutes les informations sur le matériel que YaST peut obtenir sur votre machine. Quand elle est appelée, le programme de détection du matériel est lancé. Selon votre système, cela peut prendre un certain temps. Choisissez parmi les objets de la liste et cliquez *Details* pour voir les informations détaillées sur ceux-ci. Utilisez *Sauvegarder* pour garder sur un fichier la liste détaillée sur un fichier du disque locale ou d'une disquette. Les utilisateurs avancés peuvent aussi changer le paramétrage de l'ID PCI et les réglages du noyau en choisissant *Réglage noyau*.

1.12.9 *Installation depuis des images*

L'installation depuis des images accélère considérablement l'installation. Les images contiennent des clichés (snapshot) de systèmes installés assorti de votre sélection

de modèle. Les packages qui ne sont pas contenus dans les images déployées seront installés de la façon habituelle.

A moins que votre sélection personnalisée de logiciel ne corresponde à aucune des images disponibles, cette fonctionnalité est *Activée* par défaut. En cas de problèmes, *Désactiver* la pour l'utilisation du debogage.

REMARQUE : Installation Time Stamps

The RPM database keeps track of the date a package was last installed or updated (check with for example `rpm -qa --last`). When installing from images, the time stamps of all packages originating from the images do not match the installation date but rather the date the image was created.

1.12.10 *Pare feu*

By default SuSEFirewall2 is enabled on all configured network interfaces. To globally disable the firewall for this computer, click on *Disable*. If the firewall is enabled, you may *Open* the SSH port in order to allow remote connections via secure shell.

1.13 Exécution de l'installation

Après avoir configurer toutes les paramètres de l'installation, cliquez *Installer* dans la fenêtre de démarrage de celle-ci. Quelques logiciels peuvent requérir une confirmation de licence. Si votre sélection de logiciels inclue de tels logiciels, des fenêtres de confirmation de licence sont affichée. Cliquez *Acceptez* pour installer le package. Quand vous n'êtes pas d'accord avec la licence, cliquez sur *Je n'accepte pas* et le package ne sera pas installé. Dans la boîte de dialogue suivante, confirmez avec *Installer* encore une fois.

L'installation prend habituellement entre 15 et 30 minutes, selon les performances de votre système et la quantité de logiciels choisis. Après avoir préparé le disque dur , et enfin avoir déposé les images d'installation, l'installation des logiciels commence. Pendant cette procédure un diaporama introduit les fonctionnalités de openSUSE. Choisir *Details* pour passer au log de l'installation ou *Notes de versions* pour lire les informations sur les mises à jour importantes qui n'était pas encore disponibles au moment de la parution de ce manuel..

REMARQUE : Notes de versions

The release notes that can be viewed during this step are the ones printed on the installation CD. A newer version may be available on the Internet. When manually configuring network and Internet access as described in Section 1.14.2.2, « Configuration du réseau » (p. 32), the latest version of the release notes will be displayed at the end of the installation.

Après que l'installation des logiciels est terminée, le système de base est paramétré. Entre autres, « Finir l'installation de base » inclue l'installation du gestionnaire de démarrage, l'initialisation des polices et autres. Puis YaST démarre dans le nouveau système Linux pour démarrer la configuration du système.

ASTUCE : Clefs SSH existantes

Si vous installez openSUSE sur une machine qui contient déjà des installations Linux, le programme d'installation importera automatiquement les clefs SSH d'accès le plus récent depuis l'installation existante.

1.14 Configuration du système installé.

Le système est maintenant installé, mais pas encore configuré pour l'utilisation. Le matériel, le réseau et d'autres services ne sont pas encore paramétrés. Si vous suivez la procédure d'installation par défaut, le système sera automatiquement configuré. Si vous avez désélectionné la *Configuration automatique*, la configuration manuelle du système débute.

1.14.1 Configuration Automatique du Système

Ayant redémarré, le système commence la configuration automatique. Cette routine tente de configurer votre réseau et accès internet et de paramétrer votre matériel. Ce processus ne nécessite aucune interaction. Vous pouvez changer les réglages effectués par la Configuration Automatique à tout moment sur le système installé avec YaST. Continuez avec Section 1.15, « Login Graphique » (p. 36).

1.14.2 Configuration du système Manuelle

Ayant redémarré, le système commence la configuration manuelle. Si la configuration échoue à l'une des étapes de cette phase, il redémarre et continue depuis la dernière étape menée à bien.

1.14.2.1 Nom d'hôte et nom de domaine

Le nom d'hôte est le nom de l'ordinateur sur le réseau. Le nom de domaine et le nom du réseau. Un nom d'hôte et un nom de domaine sont proposés par défaut. Si votre système fait parti d'un réseau, le nom d'hôte doit être unique sur ce réseau, tandis que le nom de domaine est commun à tous les hôtes du réseau.

In many networks, the system receives its name over DHCP. In this case it is not necessary to modify the proposed hostname and domain name. Select *Change Hostname via DHCP* instead. To be able to access your system using this hostname, even when it is not connected to the network, select *Assign Hostname to Loopback IP*. Do not enable this option when your machine provides network services. If you often change networks without restarting the desktop environment (e.g. when switching between different WLANs), do not enable this option either, because the desktop system may get confused when the hostname in `/etc/hosts` changes.

To change hostname settings at any time after installation, use *YaST Network Devices > Network Settings*. For more information, see Section 13.4.1, « Configuring the Network Card with YaST » (p. 237).

1.14.2.2 Configuration du réseau

Si vous installez openSUSE sur un PC portable, *Interfaces Contrôlées par NetworkManager* est activé. NetworkManager est un outil qui permet une connexion automatique avec un minimum d'intervention de la part de l'utilisateur. Il est idéal pour les WAN et les appareils mobiles. Si vous souhaitez utiliser la méthode traditionnelle sans NetworkManager, cliquez *Désactiver NetworkManager*. Vous trouverez des informations détaillées sur NetworkManager dans le Chapitre 25, *Using NetworkManager* (p. 437). Si vous installez openSUSE sur tout autre type de machine, la méthode traditionnelle sans NetworkManager est choisie par défaut. Cette étape de configuration vous permet de plus de configurer les périphériques réseau de

vosre système et faire des réglages concernant la sécurité, par exemple de pare-feu ou de proxy.

Le réseau peut être aussi configuré après l'installation complète du système. Si vous sautez cette étape, votre système ne sera cependant pas en mesure de récupérer les mises à jour disponibles. Pour configurer votre connexion réseau plus tard, choisissez *Skip Configuration* et cliquez *Next*.

Les réglages réseau suivant peuvent être configurés à cette étape :

Configuration Générale du Réseau

Activez ou désactivez l'utilisation de NetworkManager comme décrit ci-dessous. Le support de IPv6 peut être chagé ici. Par défaut le support de IPv6 est activé. Pour le désactiver, cliquez *Désactiver IPv6*. Pour plus d'informations sur IPv6, voir Section 13.2, « IPv6—The Next Generation Internet » (p. 225).

Pare feu

By default SuSEFirewall2 is enabled on all configured network interfaces. To globally disable the firewall for this computer, click on *Disable*. If the firewall is enabled, you may *Open* the SSH port in order to allow remote connections via secure shell. To open the detailed firewall configuration dialog, click on *Firewall*. See section « Configuring the Firewall with YaST » (Chapitre 13, *Masquerading and Firewalls*, ↑Security Guide) for detailed information.

Intefaces Réseau

Toutes les cartes réseau détectées par YaST sont listées ici. Si vous avez déjà configuré une connexion réseau pendant l'installation (comme décrit dans la Section 1.7.1.1, « Configuration Réseau » (p. 16)) la carte est utilisée pour cette connexion est listée comme *Configurée*. Un click sur *Interfaces Réseau* ouvre la boîte de dialogue, où vous pouvez changer les réglages existants, paramétrer les interfaces qui ne sont pas encore configurées, ou ajouter et configurer les cartes additionnelles.

Connexions DSL, Adapateurs ISDN, et Modems

If your computer is equipped with an internal DSL modem, an internal ADSL Fritz Card, an ISDN card or a modem, clicking on the respective headline opens the configuration dialog.

Administration à Distance par VNC

Pour activer l'administration à distance de votre machine via VNC, cliquez *Administration à Distance par VNC*. Choisir *Autoriser l'Administration à Distance* dans

la boîte de dialogue qui apparaît et ajustez les réglages correspondants du pare-feu.

Proxy

Si vous avez un serveur proxy contrôlant l'accès à Internet dans votre réseau, configurez les URL du proxy et les détails de l'authentification dans ce dialogue.

ASTUCE : Remettre à zéro la Configuration du Réseau.

Reset the network settings to the original proposed values by clicking *Change > Reset to Defaults*. This discards any changes made.

Tester la Connexion Internet

Après avoir configuré une connexion réseau, vous pouvez la tester. Pour cela, YaST établit une connexion au serveur openSUSE et télécharge les dernières notes de version. Lisez les à la fin du processus d'installation. Un test réussi est aussi une condition préalable pour l'ajout automatique des dépôts par défaut et pour la mise à jour en ligne.

Si vous avez des interfaces réseau multiples, vérifiez que la carte souhaitée est bien utilisée pour la connexion à internet. Si ce n'est pas le cas, cliquez sur *Changer de périphérique*.

Pour commencer le test, sélectionner *Oui, Tester la Connexion à Internet* et cliquez *Suivant*. Dans le dialogue qui suit, vous pouvez voir la progression du test et ses résultats. Des informations détaillées sur le processus de test est disponible via *Voir les Logs*. Si le test échoue, cliquez *Précédent* pour retourner à la configuration du réseau pour corriger votre configuration.

Continuez avec *Suivant*. Si le test est réussi, le dépôt officiel de openSUSE et la mise à jour du dépôt seront configurés. Le téléchargement des données concernant le dépôt peut prendre un certains temps la première fois.

Si vous ne voulez pas tester le connexion à cette étape, sélectionnez *Non, Passer le Test* puis *Suivant*. Cela sautera aussi le téléchargement des notes de version, et mise à jour en ligne. Ces étapes peuvent être effectuées à tout moment après la configuration initiale du système.

1.14.2.3 Mise à jour en ligne

Si une connexion internet a été établie, et les mises à jour sont disponibles, indiquez si vous souhaitez effectuer une mise à jour YaST en ligne. Si il y a plusieurs packages patchés disponibles sur le serveur, téléchargez et installez les dès que possible afin de corriger des bugs connus ou faire des mises à jour de sécurité. Pour des instructions détaillées voir le Chapitre 6, *YaST Online Update* (↑Start-Up). Les directives sur comment effectuer une mise à jour en ligne dans le système installé sont disponible à la section « Keeping the System Up-to-date » (Chapitre 5, *Installation et Désinstallation de Logiciels*, ↑Start-Up) ou le Chapitre 6, *YaST Online Update* (↑Start-Up). Cette étape est passée si aucune mise à jour n'est disponible ou si aucune connexion internet n'a été établie. Cliquez *Accepter* pour les installer et *Suivant* pour procédez à la configuration du système.

IMPORTANT : Téléchargement des mises à jour logicielles.

Le téléchargement des mises à jour peuvent prendre un certain temps, selon la bande passante de votre connexion internet et la taille des fichiers de mise à jour. Au cas où le chemin système lui même est mis à jour, la mise à jour elle même sera redémarrée et téléchargera des patch supplémentaires après le redémarrage. Si le noyau a été mis à jour, le système redémarrera avant de terminer la configuration.

1.14.2.4 Nouvel utilisateur local

If no local user was created in step one, you can create one in this dialog. To create more users, manage groups, modify defaults for new users and set up network authentication, launch *User Management*. Refer to Chapitre 10, *Managing Users with YaST* (↑Start-Up) for more information about user management. To skip this step, click *Next* without entering any data.

1.14.2.5 Notes de versions

After completing the user authentication setup, YaST displays the release notes. Reading them is recommended, because they contain important up-to-date information which was not available when the manuals were printed. If you successfully tested the Internet connection, read the most recent version of the release notes, as fetched from openSUSE's servers. Use *Miscellaneous > Release Notes* in YaST or start the SUSE Help Center to view the release notes after installation.

1.14.2.6 Configuration Matérielle

At the end of the installation, YaST opens a dialog for the configuration of *Graphics Cards Printer*. Click the individual components to start the hardware configuration. For the most part, YaST detects and configures the devices automatically.

Vous pouvez passer la configuration des périphériques pour la faire plus tard, de la façon décrite dans le Chapitre 13, *Setting Up Hardware Components with YaST* (↑Start-Up). Pour ce faire, sélectionnez *Passer la Configuration* et cliquez *Suivant*.

ASTUCE : Rétablir les paramètres par défauts de la configuration matérielles.

You can cancel any changes to the hardware configuration by clicking *Change > Reset to Defaults*. YaST then shows the original proposal again.

1.14.2.7 Finalisation de l'Installation

Après une installation réussie, YaST montre la fenêtre Installation terminée. Dans ce dialogue, vous avez la possibilité de cloner votre système, sélectionnez *Cloner votre Système pour AutoYaST*. Le profil du système actuel est stocké dans `/root/autoyast.xml`.

AutoYaST est un système pour l'installation de un ou plusieurs systèmes openSUSE automatiquement sans l'intervention de l'utilisateur. Les installations avec AutoYaST sont exécutés en utilisant un fichier de contrôle contenant les données d'installation et de configuration. Finissez l'installation de openSUSE avec *Finir* dans la fenêtre de dialogue finale.

1.15 Login Graphique

openSUSE est à présent complètement installée et configurée. Sauf si vous avez activé le login automatique ou customisé le niveau d'exécution par défaut, vous devez voir la fenêtre de login sur votre écran dans laquelle entrer votre nom d'utilisateur et votre mot de passe afin de vous connecter à votre système. Sur un système simple utilisateur avec le login automatique activé, le bureau démarre automatiquement.

Pour une courte introduction aux environnements de bureau KDE ou GNOME, référez-vous aux articles Chapitre 3, *GNOME Quick Start* (↑Start-Up) et Chapitre 2, *KDE Quick Start* (↑Start-Up). Ces manuels peuvent être lu à partir de l'*Aide* dans KDE et GNOME.

Installation à distance

openSUSE® peut être installée de différentes manières. Hormis, l'installation habituelle évoquée dans Chapitre 1, *Installation avec YaST* (p. 3), Vous pouvez faire le choix de plusieurs approches basées sur le réseau ou même opter pour une installation d'openSUSE sans aucune intervention de l'utilisateur.

Chaque méthode est introduite au moyen de deux courtes listes de vérifications : la première énumère les conditions préalables pour la méthode en question et l'autre indique la procédure de base à effectuer. Plus de détails sont fournis pour toutes les techniques utilisées dans ces scénarios d'installation.

REMARQUE

Dans les sections suivantes, le système destiné à accueillir la nouvelle installation d'openSUSE est appelé *le système cible* ou *cible d'installation*. Le terme *dépôt* (précédemment appelé « source d'installation ») est utilisé pour toutes les sources des données d'installation. Ce terme inclut les lecteurs physiques, tels que CD et DVD, ainsi que les serveurs distribuant les données d'installation sur votre réseau.

2.1 Scénarios d'Installation à Distance

Cette section introduit aux scénarios d'installation à distance les plus courants. Pour chaque scénario, vérifiez avec attention la liste des conditions préalables et suivez la

procédure indiquée dans le scénario. Si vous avez besoin d'instructions détaillées pour une étape particulière, suivez les liens indiqués dans chacun d'eux.

IMPORTANT

The configuration of the X Window System is not part of any remote installation process. After the installation has finished, log in to the target system as `root`, enter `telinit 3`, and start `SaX2` to configure the graphics hardware .

2.1.1 Simple Installation à Distance avec VNC—Configuration Réseau Statique

Ce type d'installation nécessite un minimum d'accès physique vers le système cible pour démarrer l'installation. L'installation proprement dite est entièrement contrôlée par une station de travail distante utilisant VNC pour connecter le programme d'installation. Une interaction avec l'utilisateur est cependant requise de la même manière que l'installation manuelle évoquée dans le chapitre 1, *Installation avec YaST* (p. 3).

Pour ce type d'installation assurez-vous que les conditions suivantes soit réunies :

- Un dépôt distant : NFS, HTTP, FTP ou SMB avec une connexion au réseau fonctionnelle.
- Un système cible avec une connexion réseau fonctionnelle.
- Un système de contrôle avec une connexion réseau fonctionnelle et disposant de l'installation d'un logiciel de visionnage VNC (Viewer) ou d'un navigateur avec Java (Firefox, Konqueror, Internet Explorer, Opera, etc...)
- Un support physique de démarrage (CD,DVD ou clé USB) pour chaque système cible.
- Des adresses IP statiques valides déjà assignées au dépôt et au système de contrôle.
- Une adresse IP statique valide à assignée à chaque système cible.

Pour réaliser ce type d'installation, procédez comme suit :

- 1 Paramétrez le dépôt comme indiqué dans la , Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46). Choisir un serveur NFS, HTTP ou FTP. Pour un dépôt SMB référez-vous à Section 2.2.5, « Gestion d'un dépôt SMB » (p. 54)
- 2 Démarrer le système cible en utilisant un support de démarrage (DVD,CD, clé USB) du kit média d'openSUSE . Pour des informations supplémentaires sur le kit média d'openSUSE voir Section 1.1, « Choix du Média d'installation » (p. 3).
- 3 Quand l'écran de démarrage apparaît, utiliser les options de démarrage rapides pour paramétrer les options VNC appropriées et l'adresse du dépôt. Cette étape est décrite en détails dans Section 2.4, « Démarrage du Système Cible pour l'Installation » (p. 66)

Le système cible démarre dans un environnement texte, donnant l'adresse réseau et le numéro d'affichage sous lequel l'environnement graphique d'installation peut être adressé par toute application de visualisation VNC ou navigateur. Les installations VNC s'annoncent via OpenSLP et si le firewall le permet, elle peuvent être trouvées avec Konqueror grâce au mode `service:/` ou `slp:/`.

- 4 Sur la station de contrôle, ouvrez une vue VNC ou un navigateur et connectez vous au système cible comme décrit dans Section 2.5.1, « Installation par VNC » (p. 70).
- 5 Réalisez l'installation telle qu'elle est décrite dans le Chapitre 1, *Installation avec YaST* (p. 3). Reconnectez-vous au système cible après son redémarrage pour finaliser l'installation.
- 6 Terminer l'installation.

2.1.2 Simple Installation à Distance avec VNC— Configuration Réseau Dynamique

Ce type d'installation nécessite encore un certain degré d'accès physique vers le système cible pour démarrer l'installation. Ici la configuration du réseau se fera par DHCP. L'installation elle-même est entièrement contrôlée par une station de travail distante utilisant VNC pour se connecter au programme d'installation, mais requiert cependant une interaction avec l'utilisateur pour les tâches de configuration.

Pour ce type d'installation assurez-vous que les conditions suivantes soit réunies :

- Un dépôt distant : NFS, HTTP, FTP ou SMB avec une connexion au réseau fonctionnelle.
- Un système cible avec une connexion réseau fonctionnelle.
- Un système de contrôle avec une connexion réseau fonctionnelle et disposant d'une installation du logiciel VNC Viewer ou d'un navigateur avec Java activé (Firefox, Konqueror, Internet Explorer ou Opera).
- Démarrer le système cible en utilisant un support de démarrage (DVD,CD, clé USB) du kit média d'openSUSE . Pour des informations supplémentaires sur le kit média d'openSUSE voir Section 1.1, « Choix du Média d'installation » (p. 3).
- Un serveur DHCP fonctionnel distribuant les adresses IP.

Pour réaliser ce type d'installation, procédez comme suit :

- 1** Paramétrez le dépôt comme indiqué dans la , Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46). Choisir un serveur NFS, HTTP ou FTP. Pour un dépôt SMB référez-vous à Section 2.2.5, « Gestion d'un dépôt SMB » (p. 54)
- 2** Démarrer le système cible en utilisant un support de démarrage (DVD,CD, clé USB) du kit média d'openSUSE . Pour des informations supplémentaires sur le kit média d'openSUSE voir Section 1.1, « Choix du Média d'installation » (p. 3).
- 3** Quand l'écran de démarrage apparaît, utiliser les options de démarrage rapides pour paramétrer les options VNC appropriées et l'adresse du dépôt. Cette étape est décrite en détails dans Section 2.4, « Démarrage du Système Cible pour l'Installation » (p. 66)

Le système cible démarre dans un environnement texte, donnant l'adresse réseau et le numéro d'affichage sous lequel l'environnement graphique d'installation peut être adressé par toute application de visualisation VNC ou navigateur. Les installations VNC s'annoncent via OpenSLP et si le firewall le permet, elle peuvent être trouvées avec Konqueror grâce au mode `service:/` ou `slp:/`.

- 4** Sur la station de contrôle, ouvrez une vue VNC ou un navigateur et connectez vous au système cible comme décrit dans Section 2.5.1, « Installation par VNC » (p. 70).

- 5 Réalisez l'installation telle qu'elle est décrite dans le Chapitre 1, *Installation avec YaST* (p. 3). Reconnectez-vous au système cible après son redémarrage pour finaliser l'installation.
- 6 Terminer l'installation.

2.1.3 Installation à Distance via VNC— PXE Démarrage et Réveil par le LAN (Réseau Local)

Ce type d'installation se fait sans aucune manipulation de l'utilisateur. La machine cible est allumée et démarrée à distance. L'interaction de l'utilisateur est nécessaire que lors de l'installation finale. Cette installation convient au déploiement sur des sites distants.

Pour réaliser ce type d'installation assurez vous que les conditions suivantes soient réunies :

- Un dépôt distant : NFS, HTTP, FTP ou SMB avec une connexion au réseau fonctionnelle.
- Serveur TFTP.
- Serveur DHCP Fonctionnel.
- Système cible capable de réaliser démarrage PXE, mise en réseau, et réveil par le LAN, branchés et connectés au réseau.
- Un système de contrôle avec une connexion réseau fonctionnelle et disposant d'une installation du logiciel VNC Viewer ou d'un navigateur avec Java activé (Firefox, Konqueror, Internet Explorer ou Opera).

Pour réaliser ce type d'installation, effectuez les opérations suivantes :

- 1 Paramétrer le dépôt comme indiqué dans Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46). Choisir un serveur NFS, HTTP ou FTP, ou configurer un dépôt SMB de la façon décrite dans Section 2.2.5, « Gestion d'un dépôt SMB » (p. 54).

- 2 Mettre en place un serveur TFTP de telle manière qu'il accueille une image de démarrage qui peut être téléchargée par le système cible. Ceci est décrit dans "sec.deployment.remoteinst.boot.tftp"/>.
- 3 Paramétrer le serveur DHCP pour qu'il offre des adresses IP à toutes les machines et révèle celle du serveur TFTP aux systèmes cibles. Ceci est décrit dans la section Section 2.3.1, « Configuration d'un serveur DHCP » (p. 56).
- 4 Préparer le système cible pour un démarrage PXE. Ceci est décrit dans la Section 2.3.5, « Préparation du système cible pour un boot PXE » (p. 65).
- 5 Initier le processus de démarrage du système cible en utilisant le processus de réveil par le LAN. Voir la Section 2.3.7, « Réveil par le Réseau Local (Wake on LAN) » (p. 66).
- 6 Sur la station de contrôle, ouvrez une vue VNC ou un navigateur et connectez-vous au système cible comme décrit dans Section 2.5.1, « Installation par VNC » (p. 70).
- 7 Réalisez l'installation telle qu'elle est décrite dans le Chapitre 1, *Installation avec YaST* (p. 3). Reconnectez-vous au système cible après son redémarrage pour finaliser l'installation.
- 8 Terminer l'installation.

2.1.4 Simple Installation à Distance via SSH—Configuration Statique du Réseau

Ce type d'installation requiert encore une fois un certain degré d'accès physique sur le système cible pour démarrer l'installation et pour déterminer l'adresse IP du système cible à installer. L'installation elle-même est entièrement contrôlée depuis une station distante utilisant SSH pour se connecter au programme d'installation. L'intervention de l'utilisateur est requise de la même manière que l'installation normale décrite dans Chapitre 1, *Installation avec YaST* (p. 3)..

Pour ce type d'installation assurez-vous que les conditions suivantes soient réunies :

- Un dépôt distant : NFS, HTTP, FTP ou SMB avec une connexion au réseau fonctionnelle.

- Un système cible avec une connexion réseau fonctionnelle.
- Un système de contrôle avec une connexion réseau et un client SSH.
- Démarrer le système cible en utilisant un support de démarrage (DVD,CD, clé USB) du kit média d'openSUSE . Pour des informations supplémentaires sur le kit média d'openSUSE voir Section 1.1, « Choix du Média d'installation » (p. 3).
- Des adresses IP statiques valides déjà assignées au dépôt et au système de contrôle.
- Une adresse IP statique valide à assigner à chaque système cible.

Pour réaliser ce type d'installation, procédez comme suit :

- 1** Paramétrez le dépôt comme indiqué dans la , Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46). Choisir un serveur NFS, HTTP ou FTP. Pour un dépôt SMB référez-vous à Section 2.2.5, « Gestion d'un dépôt SMB » (p. 54)
- 2** Démarrer le système cible en utilisant un support de démarrage (DVD,CD, clé USB) du kit média d'openSUSE . Pour des informations supplémentaires sur le kit média d'openSUSE voir Section 1.1, « Choix du Média d'installation » (p. 3).
- 3** Quand l'écran de démarrage apparaît, utilisez les options de démarrage rapide pour paramétrer la connexion au réseau, l'adresse du dépôt et l'activation de SSH. Cette étape est décrite en détail dans la Section 2.4.2, « Utilisation d'Options de Démarrage Personnalisées » (p. 67).

Le système cible démarre dans un environnement texte, donnant l'adresse réseau sous laquelle l'environnement graphique d'installation peut être adressé par tout client SSH.

- 4** Sur la station de contrôle, ouvrir une fenêtre de terminal et connectez-vous au système cible comme décrit dans la Section 2.5.2.2, « Connexion au programme d'installation. » (p. 72).
- 5** Réalisez l'installation telle qu'elle est décrite dans le Chapitre 1, *Installation avec YaST* (p. 3). Reconnectez-vous au système cible après son redémarrage pour finaliser l'installation.
- 6** Terminer l'installation.

2.1.5 Simple Installation à Distance via SSH—Configuration Dynamique du Réseau

This type of installation still requires some degree of physical access to the target system to boot for installation and determine the IP address of the installation target. The installation itself is entirely controlled from a remote workstation using SSH to connect to the installer, but still requires user interaction for the actual configuration efforts.

Pour ce type d'installation assurez-vous que les conditions suivantes soit réunies :

- Un dépôt distant : NFS, HTTP, FTP ou SMB avec une connexion au réseau fonctionnelle.
- Un système cible avec une connexion réseau fonctionnelle.
- Un système de contrôle avec une connexion réseau et un client SSH.
- Un support physique de démarrage (CD,DVD ou clé USB) pour chaque système cible.
- Un serveur DHCP fonctionnel distribuant les adresses IP.

Pour réaliser ce type d'installation, procédez comme suit :

- 1 Paramétrer le dépôt comme décrit dans Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46). Choisir un serveur NFS, HTTP, or FTP. Pour un dépôt SMB, se référer à Section 2.2.5, « Gestion d'un dépôt SMB » (p. 54).
- 2 Démarrer le système cible en utilisant un support de démarrage (DVD,CD, clé USB) du kit média d'openSUSE . Pour des informations supplémentaires sur le kit média d'openSUSE voir Section 1.1, « Choix du Média d'installation » (p. 3).
- 3 Quand l'écran de démarrage apparaît sur le système cible, utiliser les options de démarrage rapide pour passer les paramètres appropriés pour établir la connexion au réseau, localiser les sources d'installation, et activer SSH. Voir Section 2.4.2,

« Utilisation d'Options de Démarrage Personnalisées » (p. 67) pour des instructions détaillées sur l'utilisation de ces paramètres.

Le système cible démarre dans un environnement texte donnant l'adresse réseau sous laquelle l'environnement d'installation graphique peut-être atteint par tout client SSH.

- 4 Sur la station de contrôle, ouvrir une fenêtre de terminal et connectez-vous au système cible comme décrit dans la Section 2.5.2.2, « Connexion au programme d'installation. » (p. 72).
- 5 Réalisez l'installation telle qu'elle est décrite dans le Chapitre 1, *Installation avec YaST* (p. 3). Reconnectez-vous au système cible après son redémarrage pour finaliser l'installation.
- 6 Terminer l'installation.

2.1.6 Installation à distance via SSH— démarrage PXE et réveil par le LAN

Ce type d'installation ne nécessite aucune intervention. Le système cible est démarré et amorcé à distance.

Pour réaliser ce type d'installation assurez vous que les conditions suivantes soient réunies :

- Un dépôt distant : NFS, HTTP, FTP ou SMB avec une connexion au réseau fonctionnelle.
- Serveur TFTP.
- Serveur DHCP fonctionnel pour votre réseau, fournissant les adresses aux hôtes à installer.
- Système cible capable de réaliser démarrage PXE, mise en réseau, et réveil par le LAN, branchés et connectés au réseau.
- Un système de contrôle avec une connexion fonctionnelle et un client SSH.

Pour réaliser ce type d'installation, effectuez les opérations suivantes :

- 1 Paramétrer le dépôt comme décrit dans la Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46). Choisir un serveur NFS, HTTP ou FTP. Pour la configuration d'un dépôt SMB, se référer à la Section 2.2.5, « Gestion d'un dépôt SMB » (p. 54).
- 2 Mettre en place un serveur TFTP de telle manière qu'il accueille une image de démarrage qui peut être téléchargée par le système cible. Ceci est décrit dans `"sec.deployment.remoteinst.boot.tftp"/>`.
- 3 Paramétrer le serveur DHCP pour qu'il offre des adresses IP à toutes les machines et révèle celle du serveur TFTP aux systèmes cibles. Ceci est décrit dans la section Section 2.3.1, « Configuration d'un serveur DHCP » (p. 56).
- 4 Préparer le système cible pour un démarrage PXE. Ceci est décrit dans la Section 2.3.5, « Préparation du système cible pour un boot PXE » (p. 65).
- 5 Initier le processus de démarrage du système cible en utilisant le processus de réveil par le LAN. Voir la Section 2.3.7, « Réveil par le Réseau Local (Wake on LAN) » (p. 66).
- 6 Sur la station de contrôle démarrer un client SSH et le connecter au système cible comme décrit dans Section 2.5.2, « Installation SSH » (p. 72).
- 7 Réalisez l'installation telle qu'elle est décrite dans le Chapitre 1, *Installation avec YaST* (p. 3). Reconnectez-vous au système cible après son redémarrage pour finaliser l'installation.
- 8 Terminer l'installation.

2.2 Configuration du serveur contenant les sources de l'installation.

Selon le système d'exploitation exécuté par la machine utilisée comme source d'installation dans le réseau pour openSUSE plusieurs options sont disponibles pour la configuration du serveur. La manière la plus facile pour configurer un

serveur d'installation est celle utilisant YaST voir openSUSE pour la version 11.1 et supérieure.

ASTUCE

Vous pouvez aussi utiliser un serveur Microsoft Windows comme serveur d'installation pour le déploiement d'un Linux. Voir la Section 2.2.5, « Gestion d'un dépôt SMB » (p. 54) pour les détails.

2.2.1 Configuration et installation d'un serveur YaST

YaST offre un outil graphique pour la création d'un dépôt sur un réseau. Il supporte des serveurs d'installation HTTP, FTP, et NFS.

- 1 Se connecter en tant que `root` à la machine destinée à être le serveur d'installation.
- 2 Installer le package `yast2-instserver`
- 3 Start *YaST* > *Miscellaneous* > *Installation Server*.
- 4 Choisir le type de dépôt (HTTP, FTP ou NFS). Le service choisi est démarré automatiquement à chaque démarrage du système. Si un service du type sélectionné est déjà en cours sur votre système et que vous voulez configurer celui-ci manuellement pour le serveur, désactivez la configuration automatique du service avec *Ne configurer aucun des services réseau*. Dans les deux cas indiqué le répertoire du serveur dans lequel vous souhaitez mettre à disposition les données d'installation.
- 5 Configurer le type de dépôt nécessaire. Cette étape concerne la configuration automatique des services du serveur. Elle est sautée quand la configuration automatique est désactivée.

Définir un alias pour le répertoire racine du serveur FTP ou HTTP sur lequel les données d'installation doivent être trouvées. Le dépôt sera ensuite localisé sous `ftp://IPduServeur/Alias/Nom` (HTTP). Où *Nom* est le nom du dépôt, tel qu'il a été défini dans les étapes précédentes. Si vous avez choisi NFS dans les étapes précédentes, définir les jokers (wild cards) et les options d'exportation.

Le serveur NFS sera accessible sous `nfs://IPduServeur/Nom`. Des détails sur les exportations dans NFS peuvent être trouvés dans Chapitre 18, *Sharing File Systems with NFS* (p. 325).

ASTUCE : Paramétrage du Firewall.

Assurez-vous que les réglages du pare-feu de votre serveur permettent le trafic sur les ports HTTP, NFS et FTP. Si ces permissions ne sont pas activées *ouvrir les ports du pare-feu* ou vérifier *Détails du pare-feu*.

- 6 Configurer le dépôt. Avant que le média d'installation ne soit copié vers sa destination, définir un nom pour le dépôt (de préférence une abréviation facile à mémoriser du produit et de sa version). YaST permet de fournir des images ISO au lieu de copies des DVD d'installations. Si vous désirez utiliser des images ISO, cocher la case correspondante et spécifiez le chemin d'accès du répertoire dans lequel les fichiers ISO peuvent être trouvés localement. Selon le produit à distribuer par l'intermédiaire du dépôt d'installation, il se pourrait que soient nécessaires à l'installation, des CD supplémentaires ou de services pack. Ils devront être alors, ajoutés comme dépôts supplémentaires. Pour annoncer le serveur d'installation sur le réseau via OpenSLP, activez l'option appropriée.

ASTUCE

Envisagez l'annonce de votre dépôt via OpenSLP si votre configuration réseau supporte cette option. Cela vous évite d'entrer dans le chemin d'installation réseau sur chaque machine cible. Le système cible sera alors démarré en utilisant les options SLP de démarrage et trouvera le dépôt sur le réseau sans aucune autre configuration supplémentaire. Pour des détails sur cette option référez-vous à Section 2.4, « Démarrage du Système Cible pour l'Installation » (p. 66).

- 7 Télécharger vers le serveur les données d'installation. L'étape la plus longue dans la configuration d'un serveur d'installation est la copie du/des média(s) d'installation. Insérez les médias dans l'ordre demandé par YaST et attendez la fin de la copie. Quand les sources ont été complètement copiées, retournez à la vue d'ensemble des dépôts existant et fermez la fenêtre de configuration en cliquant sur *Fin*.

Votre serveur d'installation est maintenant totalement configuré et prêt pour l'installation à distance. Il sera automatiquement activé toutes les fois où le système démarrera. Aucune intervention supplémentaire n'est requise. Vous aurez seule-

ment besoin de configurer et démarrer le service manuellement si vous avez choisi de désactiver la configuration automatique du service réseau depuis YaST telle qu'indiquée dans l'étape initiale.

Pour désactiver un dépôt , choisir le dépôt à supprimer et choisir *Effacer*. Les données d'installation seront supprimer du système. Pour désactiver un des services réseau, utiliser le module YaST correspondant.

Si votre serveur d'installation a besoin de fournir les données d'installation pour plus d'un élément de la version du produit à installer, démarrez le module du serveur d'installation dans YaST et choisissez *Ajouter* dans la vue d'ensemble des dépôts existants pour configurer un nouveau dépôt.

2.2.2 Configurer un dépôt NFS manuellement.

Le paramétrage d'un dépôt NFS se fait en deux étapes essentielle. La première : création du répertoire contenant les données d'installation et copie des medias d'installation dans cette structure. La seconde : exportation du répertoire contenant les données vers le réseau.

Pour créer le répertoire contenant les données d'installation procédez comme suit :

- 1 Connectez-vous en tant que `root`.
- 2 Créez le répertoire amené à accueillir les données d'installation et placez-vous dans ce répertoire. Par exemple

```
mkdir install/product/productversion
cd install/product/productversion
```

Remplacez *produit* avec une abréviation du nom du produit et *version* avec une chaîne contenant le nom du produit et sa version.

- 3 Pour chaque DVD du kit média exécutez les commandes suivantes :

- 3a Copier le contenu entier du DVD dans le répertoire dédié du serveur d'installation :

```
cp -a /media/path_to_your_DVD_drive .
```

Remplacer `chemin_Vers_DVD` par le chemin indiquant celui de votre lecteur DVD. Selon le type de lecteur utilisé par votre système, ce peut être `cdrom`, `cdrecorder`, `dvd`, ou `dvdrecorder`.

3b Renommer le répertoire de chaque DVD avec son numéro :

```
mv path_to_your_DVD_drive DVDx
```

Ou x représente le numéro de DVD.

Sur openSUSE, vous pouvez exporter le dépôt avec NFS en utilisant YaST. En faisant comme suit :

- 1 Connectez-vous en tant que `root`.
- 2 Start *YaST* > *Network Services* > *NFS Server*.
- 3 Choisir *Démarrer et Ouvrir les ports du pare feu* et cliquer *Suivant*.
- 4 Choisir *Ajouter un répertoire* et naviguer vers le répertoire contenant les sources d'installation, C'est à dire *version*.
- 5 Choisir *Ajouter Hôte* et entrer le nom d'hôte de la machine vers laquelle exporter les données d'installation. Au lieu de spécifier un nom d'hôte, vous pouvez aussi utiliser un joker, une plage d'adresse réseau, ou juste le nom de domaine de votre réseau. Entrer les options d'exportation appropriées ou laisser les valeurs par défaut, qui fonctionneront sur la plupart des configurations. Pour plus d'informations sur la syntaxe utilisée pour exporter un partage NFS, lire les pages de manuel de `exports`.
- 6 Cliquer *Finir*. Le serveur NFS contenant les dépôts de openSUSE sont automatiquement activé et intégré au système pendant son démarrage.

Si vous préférez exporter manuellement le dépôt via NFS au lieu d'utiliser le module de serveur NFS dans YaST procédez comme indiqué ci-dessous :

- 1 Connectez-vous en tant que `root`.
- 2 Ouvrez le fichier `/etc/exports` et entrez la ligne suivante

```
/productversion *(ro,root_squash,sync)
```

Ceci exporte le répertoire `/version` vers tout hôte faisant partie de ce réseau ou à tout hôte pouvant se connecter à ce serveur. Pour limiter l'accès à ce serveur, utiliser les masques de sous-réseaux ou les noms de domaines au lieu des jokers `*`. Référez vous au pagres de manuel sur `export` pour les détails.

- 3 Pour ajouter un service NFS à la liste des serveurs démarrés pendant la phase de démarrage du système, exécuter les commandes suivantes :

```
insserv /etc/init.d/nfsserver
```

- 4 Démarrer le serveur NFS avec `rcnfsserver start`. Si vous avez besoin de changer la configuration de votre serveur NFS plus tard, modifier le fichier de configuration et redémarrer le demon NFS avec `rcnfsserver restart`.

L'annonce d'un serveur NFS via OpenSLP permettra aux clients du réseau de connaître l'adresse réseau du serveur.

- 1 Connectez-vous en tant que `root`.

- 2 Créer le fichier de configuration `/etc/slp.reg.d/`

`install.suse.nfs.reg` en y écrivant les lignes suivantes :

```
# Register the NFS Installation Server
service:install.suse:nfs://$HOSTNAME/path_to_repository/DVD1,en,65535
description=NFS Repository
```

Remplacez `path_to_repository` Avec le chemin réel de votre dépôt.

- 3 Démarrer le demon OpenSLP avec la commande `rcslpd start`.

Pour des informations supplémentaires sur OpenSLP, vous pouvez vous référez à la documentation du pack située dans `/usr/share/doc/packages/openslp/` ou consultez Chapitre 14, *SLP Services in the Network* (p. 279). Plus d'informations sur NFS voir Chapitre 18, *Sharing File Systems with NFS* (p. 325).

2.2.3 Configuration d'un dépôt FTP manuellement.

La création d'un dépôt FTP se fait de façon très similaires à la création d'un dépôt NFS. Un dépôt FTP peut-être annoncé sur le réseau en utilisant OpenSLP de la même manière.

- 1** Créer un répertoire contenant les sources d'installation comme décrit dans Section 2.2.2, « Configurer un dépôt NFS manuellement. » (p. 49).
- 2** Configurer le serveur FTP pour qu'il distribue le contenu de votre répertoire d'installation.
 - 2a** Se connecter en tant que `root` et installer le pack `vsftpd` avec le gestionnaire de programme de YaST.
 - 2b** Entrer dans le répertoire racine du serveur FTP:

```
cd /srv/ftp
```
 - 2c** Créer un sous-répertoire contenant les sources de l'installation dans le répertoire racine du serveur FTP.

```
mkdir repository
```

Remplacer *repository* avec le nom du produit.
 - 2d** Monter le contenu du répertoire d'installation dans l'environnement chroot du serveur FTP :

```
mount --bind path_to_repository /srv/ftp/repository
```

Remplacer *path_to_repository* et *repository* avec les valeurs correspondant à votre configuration. Si vous voulez rendre cette configuration permanente, ajoutez la à `/etc/fstab`.
 - 2e** Démarrer `vsftpd` avec la commande `vsftpd`.
- 3** Annoncer le dépôt via OpenSLP, si ceci est supporté par la configuration de votre réseau :
 - 3a** Créer le fichier de configuration `/etc/slp.reg.d/install.suse.ftp.reg` avec les lignes suivantes :

```
# Register the FTP Installation Server
service:install.suse:ftp://$HOSTNAME/repository/DVD1,en,65535
description=FTP Repository
```

Remplacer *repository* avec le nom de répertoire de votre dépôt sur le serveur. La ligne `service :` doit être entrée sous la forme d'une ligne continue.

3b Démarrer le demon OpenSLP avec la commande `rcslpd start`.

ASTUCE : Configuration d'un serveur FTP avec YaST.

Si vous préférez utiliser YaST au lieu de configurer le serveur FTP manuellement, référez vous au Chapitre 21, *Setting up an FTP server with YaST* (p. 393) Pour de plus amples informations sur la façon d'utiliser le module de serveur FTP de YaST

2.2.4 Configuration d'un dépôt HTTP manuellement.

La création d'un dépôt HTTP est très similaire a celle d'un dépôt NFS. Un dépôt HTTP peut-être annoncé sur le réseau en utilisant OpenSLP de la même manière.

- 1** Créer un répertoire contenant les sources d'installation comme décrit dans Section 2.2.2, « Configurer un dépôt NFS manuellement. » (p. 49).
- 2** Configurer le serveur HTTP pour distribuer le contenu de votre répertoire d'installation:
 - 2a** Installer Apache comme décrit dans la Section 20.1.2, « Installation » (p. 352).
 - 2b** Entrer dans le répertoire racine du serveur HTTP (`/srv/www/htdocs`) et créer le sous-répertoire qui contiendra les sources de l'installation:

```
mkdir repository
```

Remplacer *repository* avec le nom du produit.
 - 2c** Créer un lien symbolique des sources d'installation vers le répertoire racine du serveur web (`/srv/www/htdocs`):

```
ln -s /path_to_repository /srv/www/htdocs/repository
```
 - 2d** Modifier le fichier de configuration du serveur HTTP (`/etc/apache2/default-server.conf`) pour qu'il suive le lien symbolique.
Remplacer la ligne suivante :

Options None

avec

Options Indexes FollowSymLinks

2e Recharger la configuration de votre serveur HTTP en utilisant la commande `rcapache2 reload`.

3 Annoncer le dépôt via OpenSLP, si ceci est supporté par la configuration de votre réseau :

3a Créer le fichier de configuration `/etc/slp.reg.d/install.suse.http.reg` avec les lignes suivantes:

```
# Register the HTTP Installation Server
service:install.suse:http://$HOSTNAME/repository/DVD1/,en,65535
description=HTTP Repository
```

Remplacer *dépôt* par le chemin vers le dépôt de votre serveur. La ligne `service:` doit être rentré sous la forme d'une ligne continue.

3b Démarrer le demon OpenSLP en utilisant la commande `rcslpd restart`.

2.2.5 Gestion d'un dépôt SMB

En utilisant SMB vous pouvez importer les sources d'installation d'un serveur Microsoft Windows et démarrer un déploiement sans la présence de machine Linux sur le réseau.

Pour configurer un partage Windows contenant votre dépôt openSUSE procédez comme suit :

- 1** Connectez-vous à votre poste Windows.
- 2** Créer un nouveau dossier qui contiendra l'arborescence entière de l'installation et nommez le `INSTALL`, par exemple.
- 3** Exportez ce partage conformément à la procédure indiquée dans la documentation de votre Windows.

- 4 Entrez dans le partage et créer un sous-dossier, appelé *produit*. Où *produit* est le nom du produit dont vous voulez configurer le dépôt.
- 5 Entrez dans le dossier `INSTALL/produit` et y copier chaque DVD dans un dossier distinct tel que `DVD1` et `DVD2`.

Pour utiliser un partage SMB monté comme dépôt, procédez de cette façon :

- 1 Démarrer la cible d'installation.
- 2 Choisir *Installation*.
- 3 Presser F4 pour la sélection du dépôt.
- 4 Choisir SMB et entrer le nom de station Windows ou son adresse IP, le nom de partage (`INSTALL/produit/DVD1`, dans notre exemple), un nom d'utilisateur et un mot de passe.

Après avoir valider avec Entrée, YaST démarre et vous pouvez alors réaliser l'installation.

2.2.6 Utilisation des images ISO du support d'installation sur le serveur

Au lieu de copier le support physique dans le répertoire de votre serveur manuellement, vous pouvez aussi monter les images ISO des supports d'installation dans votre serveur d'installation et l'utiliser comme un dépôt. Pour paramétrer un serveur HTTP, NFS ou FTP utilisant des images ISO au lieu de copies des support, procédez comme suit :

- 1 Télécharger les images ISO et les enregistrer sur le poste utiliser comme serveur d'installation.
- 2 Connectez-vous en tant que `root`.
- 3 Choisir et créer un emplacement approprié pour les données d'installation, de la façon décrite dans la Section 2.2.2, « Configurer un dépôt NFS manuellement. » (p. 49), Section 2.2.3, « Configuration d'un dépôt FTP manuellement. » (p. 51), ou Section 2.2.4, « Configuration d'un dépôt HTTP manuellement. » (p. 53).

- 4 créer un sous-répertoire pour chaque DVD.
- 5 Pour monter et dépaquetter chaque images ISO vers sa destination finale, utiliser les commandes suivantes:

```
mount -o loop path_to_isopath_to_repository/product/mediumx
```

Où *chemin_vers_iso* est le chemin local vers la copie de l'image ISO, *chemin_dépôt* le répertoire des sources sur votre serveur, *produit* le nom du produit, et *supportx* le type de support (CD ou DVD) et son numéro.

- 6 Répéter les étapes précédentes pour monter toutes les images ISO nécessaire à l'installation.
- 7 Démarrer le serveur d'installation comme à la normale, de la façon décrite dans la Section 2.2.2, « Configurer un dépôt NFS manuellement. » (p. 49), Section 2.2.3, « Configuration d'un dépôt FTP manuellement. » (p. 51), ou Section 2.2.4, « Configuration d'un dépôt HTTP manuellement. » (p. 53).

Pour monter automatiquement une image ISO au démarrage, ajouter les entrées de montage respectives dans `/etc/fstab`. Une entrée dans l'esprit de l'exemple précédent ressemblerait à celle-ci :

```
path_to_iso path_to_repository/productmedium auto loop
```

2.3 Préparer le démarrage du système cible

Cette section couvre les tâches de configuration nécessaires au démarrage de scénarios complexes. Elle contient des exemples de configuration pré-appliqué pour les installation avec DHCP, démarrage PXE, TFTP et réveil par le réseau.

2.3.1 Configuration d'un serveur DHCP

Il y a deux manières de paramétrer un serveur DHCP. Pour openSUSE, YaST fournit une interface graphique pour cette opération. Les utilisateurs peuvent aussi éditer les fichiers de configuration du serveur. Pour plus d'informations sur DHCP voir aussi Chapitre 16, *DHCP* (p. 307).

2.3.1.1 Configuration du serveur DHCP avec YaST

Pour annoncer l'adresse du serveur TFTP au clients du réseau et spécifier une image bootable que la cible d'installation devrait utiliser, ajouter deux déclarations à la configuration de votre serveur DHCP.

- 1 Connectez-vous en tant que `root` à la machine hébergeant le serveur DHCP.
- 2 Installez le pack `yast2-dhcp-server`.
- 3 Start *YaST* > *Network Services* > *DHCP Server*.
- 4 Exécutez l'assistant pour une configuration basique du serveur DHCP.
- 5 Choisissez *Expert* puis *Oui* au moment de quitter l'écran de démarrage.
- 6 Dans la boîte de dialogue *Déclarations configurées*, sélectionnez le sous-réseau dans lequel le nouveau système doit être situé et cliquez *Modifier guimenu*.
- 7 Dans la boîte de dialogue *configuration du sous-réseau* choisissez *Ajouter* pour ajouter la nouvelle option de configuration de sous-réseau.
- 8 Choisissez `filename` et entrez `pxelinux.0` comme valeur.
- 9 Ajoutez une autre option (`next-server`) et positionnez cette valeur sur l'adresse du serveur TFTP.
- 10 Choisissez *OK* et *Finir* pour terminer la configuration du serveur DHCP.

Pour configurer DHCP de façon à ce qu'il procure une IP statique à un hôte spécifique, entrer dans *Réglages experts* du module de configuration du serveur DHCP (Étape 5 (p. 57)) et ajouter une nouvelle déclaration pour cet hôte.

2.3.1.2 Configuration manuelle d'un serveur DHCP

Les seules choses que le serveur DHCP doivent faire, à part fournir des adresses automatiques aux clients du réseaux, sont d'annoncer l'adresse du serveur TFTP et les fichiers qui ont besoin d'être chargés par les routines d'installation sur les systèmes cibles.

- 1 Connectez-vous en tant que `root` à la machine hébergeant le serveur DHCP.

2 Ajoutez les lignes suivantes au fichier de configuration de sous-réseau de votre serveur DHCP situé dans `/etc/dhcpd.conf`:

```
subnet 192.168.1.0 netmask 255.255.255.0 {
    range dynamic-bootp 192.168.1.200 192.168.1.228;
    # PXE related stuff
    #
    # "next-server" defines the tftp server that will be used
    next-server ip_tftp_server:
    #
    # "filename" specifies the pxelinux image on the tftp server
    # the server runs in chroot under /srv/tftpboot
    filename "pxelinux.0";
}
```

Remplacez *ip_of_the_tftp_server* par l'adresse IP réelle de votre serveur TFTP. Pour plus d'informations sur les options disponibles dans `dhcpd.conf`, référez-vous au manuel de `dhcpd.conf`.

3 Redémarrez le serveur DHCP en exécutant `rcdhcpd restart`.

Si vous prévoyez d'utiliser SSH pour le contrôle distant de PXE et du réveil par le réseau local (Wake on LAN), spécifiez explicitement les adresse IP fournies par DHCP aux cibles d'installation. Pour accomplir cela, modifier la configuration DHCP mentionnée plus haut en fonction de l'exemple suivant:

```
group {
    # PXE related stuff
    #
    # "next-server" defines the tftp server that will be used
    next-server ip_tftp_server:
    #
    # "filename" specifies the pxelinux image on the tftp server
    # the server runs in chroot under /srv/tftpboot
    filename "pxelinux.0";
    host test {
        hardware ethernet mac_address;
        fixed-address some_ip_address;
    }
}
```

La déclaration d'hôte introduit le nom d'hôte de la cible d'installation. Pour lier le nom d'hôte et IP à un hôte spécifique, vous devez connaître et spécifier l'adresse physique du système (adresse MAC). Remplacer toutes les variables utilisées dans cet exemple avec les valeurs réelles qui correspondent à votre environnement

Après avoir redémarré le serveur DHCP, celui-ci fournira une adresse statique à l'hôte spécifié, vous permettant ainsi de vous connecter à celui-ci via SSH.

2.3.2 Configuration d'un serveur TFTP

Configurez un serveur TFTP avec YaST ou configurez le manuellement sur tout autre système d'exploitation Linux supportant xinetd et TFTP. Le serveur TFTP délivrera l'image de démarrage au système cible dès que ce dernier démarrera et enverra une requête au serveur pour cela.

2.3.2.1 Configuration d'un serveur TFTP avec YaST

- 1 Connectez-vous en tant que `root`.
- 2 Installez le paquet `yast2-tftp-server`.
- 3 Start *YaST > Network Services > TFTP Server* and install the requested package.
- 4 Cliquez *Enable* pour être sûr que le serveur est démarré et inclus dans une routine de démarrage. Aucune autre action de votre part n'est nécessaire pour garantir cela. xinetd démarrera tftpd au moment du démarrage.
- 5 Cliquez *Open Port in Firewall* pour ouvrir les ports appropriés dans le firewall en place sur votre station. S'il n'y a pas de pare-feu sur votre serveur cette action est inutile.
- 6 Cliquez *Parcourir* pour naviguer dans le répertoire de l'image de démarrage. Le répertoire par défaut `/tftpboot` est créé et choisi automatiquement.
- 7 Cliquez *Finir* pour appliquer vos réglages et démarrer le serveur.

2.3.2.2 Configuration Manuelle d'un serveur TFTP.

- 1 Connectez-vous en tant que `root` et installez les paquets `tftp` et `xinetd`.
- 2 S'il ne sont pas disponibles, créez les répertoires `/srv/tftpboot` et `/srv/tftpboot/pxelinux.cfg`.
- 3 Ajoutez les fichiers appropriés nécessaires à l'image de démarrage comme décrit dans Section 2.3.3, « Utiliser le démarrage PXE. » (p. 60).
- 4 Modifiez la configuration de `xinetd` situées sous `/etc/xinetd.d` pour s'assurer que le serveur TFTP soit activé au démarrage:

4a Si celui-ci n'existe pas, créer le fichier appelé `tftp` dans ce répertoire avec `touch tftp`. Puis exécutez `chmod 755 tftp`.

4b Ouvrir le fichier `tftp` et y ajouter les lignes suivantes :

```
service tftp
{
    socket_type          = dgram
    protocol             = udp
    wait                 = yes
    user                 = root
    server               = /usr/sbin/in.tftpd
    server_args          = -s /srv/tftpboot
    disable              = no
}
```

4c Sauvegardez le fichier et redémarrer `xinetd` avec `rcxinetd restart`.

2.3.3 Utiliser le démarrage PXE.

Some technical background information as well as PXE's complete specifications are available in the Preboot Execution Environment (PXE) Specification (<http://www.pix.net/software/pxeboot/archive/pxespec.pdf>).

1 Accédez au répertoire `boot/<architecture>/loader` de votre dépôt d'installation et copiez les fichiers `linux`, `initrd`, `message`, `biostest`, et `memtest` dans le répertoire `/srv/tftpboot` en entrant les commandes suivantes :

```
cp -a linux initrd message biostest memtest /srv/tftpboot
```

2 Installez le paquet `syslinux` directement depuis le DVD d'installation avec YaST.

3 Copiez le fichier `/usr/share/syslinux/pxelinux.0` dans `/srv/tftpboot` en entrant les commandes :

```
cp -a /usr/share/syslinux/pxelinux.0 /srv/tftpboot
```

4 Allez dans le répertoire de votre dépôt d'installation et copiez `isolinux.cfg` dans `/srv/tftpboot/pxelinux.cfg/default` en entrant:

```
cp -a boot/<architecture>/loader/isolinux.cfg /srv/tftpboot/pxelinux.cfg/default
```

- 5 Edit the `/srv/tftpboot/pxelinux.cfg/default` file and remove the lines beginning with `readinfo` and `framebuffer`.
- 6 Insérez les entrées suivantes dans les lignes ajoutées par défauts au labels `fail-safe` et `apic`

```
insmod=kernel module
```

Au moyen de cette entrée, entrez le module de noyau réseau nécessaire pour l'installation sur le client PXE. Remplacez `kernel module` avec le nom approprié de module pour votre périphérique réseau.

```
netdevice=interface
```

Cette entrée définit l'interface réseau du client qui doit être utilisée pour l'installation par le réseau. Elle est nécessaire seulement si le client est équipé avec plusieurs cartes réseau et doit donc être adapté en conséquence. Dans le cas d'une seule carte réseau cette entrée peut être omise.

```
install=nfs://ip_instserver/path_to_repository/DVD1
```

Cette entrée définit le serveur NFS et le dépôt pour le client à installer. Remplacer `ip_instserver` par l'adresse IP réelle du serveur d'installation. `path_to_repository` doit être remplacé par le chemin vers le dépôt. Les dépôts HTTP, FTP ou SMB sont indiqués de manières similaires, excepté pour le préfixe du protocole, qui devra être respectivement `http`, `ftp`, ou `smb`.

IMPORTANT

Si vous avez besoin de passer d'autres options de démarrage à la routine d'installation, tels que des paramètres SSH ou VNC, ajoutez ceux-ci dans l'entrée `install`. Une vue d'ensemble des paramètres et quelques exemples sont donnés dans Section 2.4, « Démarrage du Système Cible pour l'Installation » (p. 66).

ASTUCE : Modification des noms de fichiers du noyau et de `init.rd`

Il est possible d'utiliser différents noms de fichiers images du noyau et de `init.rd`. Ceci est très utile si vous voulez fournir différents systèmes d'exploitation depuis le même serveur d'installation. Cependant, vous devez faire attention à prendre en compte le fait qu'un seul point n'est

autorisé dans les noms de fichiers qui seront fournis par TFTP pour un démarrage PXE.

Dans l'exemple `/srv/tftpboot/pxelinux.cfg/default` de fichier suivant, Ajustez le préfixe du protocole pour le dépôt pour qu'il corresponde à votre configuration réseau et spécifiez la méthode choisie pour la connexion à l'installateur en ajoutant les options `vnc` et `vncpassword` ou `usessh` et `sshpassword` à l'entrée `install`. Les lignes séparées par `\` doivent être entrées sous la forme d'une ligne continue sans retour à la ligne et sans le caractère `\`.

```
default hddisk

# default
label linux
    kernel linux
    append initrd=initrd ramdisk_size=65536 \
        install=nfs://ip_instserver/path_to_repository/product/DVD1

# rescue
label rescue
    kernel linux
    append initrd=initrd ramdisk_size=65536 rescue=1

# bios test
label firmware
    kernel linux
    append initrd=biostest,initrd splash=silent install=exec:/bin/
run_biostest showopts

# memory test
label memtest
    kernel memtest

# hard disk
label hddisk
    localboot 0

implicit      0
display       message
prompt       1
timeout       100
```

- 7 Remplacer `ip_instserver` et `path_to_repository` avec les valeurs utilisées par votre configuration.

Les sections suivantes serviront de référence rapide pour les options utilisées dans cette configuration. Vous trouverez plus d'informations sur les options disponibles

dans la documentation du paquet `syslinux` située sous `/usr/share/doc/packages/syslinux/`.

2.3.4 Les Options de Configuration PXELINUX

Les options listées ci-dessous sont un sous-ensemble de toutes les options disponibles pour les fichiers de configuration PXELINUX.

`DEFAULT kernel options...`

Configure la ligne de commande passée par défaut au noyau. Si PXELINUX démarre automatiquement, il se comportera comme si les entrées après `DEFAULT` étaient saisies dans l'invite de démarrage, excepté l'option `auto` qui est automatiquement ajoutée, indiquant un démarrage automatique.

Si aucun fichier de configuration n'est présent ou qu'aucune entrée `DEFAULT` n'est présente dans le fichier de configuration, la valeur par défaut est le nom de noyau « `linux` » sans aucune option.

`APPEND options...`

Ajoute une ou plusieurs options dans la ligne de commande du noyau. Celles-ci sont ajoutées à la fois pour le démarrage automatique et manuel. Les options sont ajoutées au tout début de la ligne de commande du noyau, permettant habituellement, l'entrée d'options de noyau explicites afin de les surcharger.

`LABEL label KERNEL image APPEND options...`

Indique que si `label` est entrée comme noyau à démarré, PXELINUX devra démarré `image` et les options `APPEND` spécifiées plutôt que celles spécifiées dans la section globale du fichier (avant la première commande `LABEL`). la valeur par défaut de `image` est la même que celle de `label` et si aucune valeur `APPEND` n'est donnée, les entrées globales (s'il y en a) seront utilisées par défaut. Jusqu'à 128 entrées `LABEL` sont permises.

Notez que si GRUB utilise la syntaxe suivante :

```
title mytitle
  kernel my_kernelmy_kernel_options
  initrd myinitrd
```

PXELINUX utilise la syntaxe suivante :

```
label mylabel
```

```
kernel mykernel
append myoptions
```

Labels are mangled as if they were filenames and they must be unique after mangling. For example, the two labels « v2.6.30 » and « v2.6.31 » would not be distinguishable under PXELINUX because both mangle to the same DOS filename.

Le noyau n'est pas forcément un noyau Linux; il peut être aussi un secteur de boot ou un fichier COMBOOT.

APPEND -

Ne rien ajouter. APPEND suivi d'un simple tiret comme argument dans une section LABEL peut être utilisé pour surcharger les options globales APPEND.

LOCALBOOT *type*

Sur PXELINUX, spécifier l'option LOCALBOOT 0 au lieu de KERNEL signifie l'invocation de cette étiquette particulière et causera un démarrage sur le disque local au lieu d'un démarrage noyau.

Argument	Description
0	Effectue un démarrage normal
4	Effectue un démarrage local avec le pilote réseau universel (Universal Network Driver Interface (UNDI)) chargé en mémoire.
5	Effectue un démarrage local avec la pile entière PXE, incluant UNDI, chargée en mémoire

Toutes les autres valeurs sont indéfinies. Si vous ne savez ce que sont UNDI ou PXE, spécifiez 0.

TIMEOUT *time-out*

Indique la quantité de temps à attendre, en 1/10ème de seconde, pendant l'invite de démarrage avant un démarrage automatique. Cette limite de temps est annulée dès que l'utilisateur tape quoi que ce soit au clavier, en supposant que celui-ci termine la commande qu'il a initiée. Un time-out à zéro (c'est la valeur par

défaut) annule complètement cette limite. La valeur maximale de time-out est 35996 (un peu moins d'une heure).

PROMPT *flag_val*

Si *flag_val* est à zéro, l'invite de démarrage est affichée seulement si Maj ou Alt sont pressées ou si Caps Lock ou Scroll Lock sont activées (c'est l'option par défaut).

F2 *filename*
F1 *filename*
...etc...
F9 *filename*
F10 *filename*

Affiche les fichiers indiqués sur l'écran quand une touche fonction est pressée pendant l'invite de démarrage. Ce peut être utile pour implémenter une aide en ligne au moment du pré-démarrage (généralement pour des options en ligne de commande pour le noyau). Pour la compatibilité ascendante avec les versions plus ancienne, F0 peut se substituer à F10. Notez que il n'y a par contre aucun moyen de lier les touches F11 et F12 à des noms de fichiers.

2.3.5 Préparation du système cible pour un boot PXE

Préparez le BIOS pour un démarrage PXE en incluant l'option PXE à la liste de gestion des priorités de démarrage des périphériques (boot order).

AVERTISSEMENT : BIOS Boot Order (ordre de démarrage)

Ne placez pas l'option PXE devant le disque dur local dans le BIOS. Sinon le système essaiera de se réinstaller lui même à chaque démarrage.

2.3.6 Préparer le système cible pour un réveil par le LAN

Wake on LAN (WOL) (Réveil par le Réseau Local) requiert que les options appropriées soient activées avant toute installation. Notez aussi, l'adresse MAC du système cible, car celle-ci est nécessaire pour initier le réveil par le LAN.

2.3.7 Réveil par le Réseau Local (Wake on LAN)

Le réveil par le réseau local (Wake on LAN) permet à une machine d'être allumée à distance en utilisant un paquet réseau contenant son adresse MAC. Comme toutes les machines de part le monde ont une adresse MAC différentes, vous n'avez donc pas à vous inquiéter du réveil accidentel de la mauvaise machine.

IMPORTANT : Wake on LAN sur des segments réseau différents

Si la machine contrôlée n'est pas située sur le même segment réseau que la cible d'installation devant être réveillée, vous devrez soit configurer les requêtes WOL pour qu'elles soient envoyées en multicasts, soit contrôler à distance une machine sur le réseau cible pour que celle-ci agisse comme envoyeur de ces requêtes.

2.4 Démarrage du Système Cible pour l'Installation

Fondamentalement, il existe deux façons de personnaliser le processus de démarrage de l'installation, mise à part celle mentionnée dans Section 2.3.7, « Réveil par le Réseau Local (Wake on LAN) » (p. 66) et Section 2.3.3, « Utiliser le démarrage PXE. » (p. 60). Vous pouvez soit utiliser les options de démarrage par défaut et les touches de fonction, soit utiliser l'invite de démarrage de l'écran d'installation pour passer toutes options dont le noyau d'installation aurait besoin pour un matériel particulier.

2.4.1 Utilisation des options de démarrage par défaut

Les options de démarrage sont décrites en détail dans Chapitre 1, *Installation avec YaST* (p. 3). Généralement, une simple sélection de *Installation* démarre de processus de démarrage.

Si des problèmes viennent à survenir, utilisez *Installation—ACPI Disabled* ou *Installation—Safe Settings*. Pour plus d'information pour le dépannage, référez-vous à section « Installation Problems » (Annexe A, *Aide et problèmes*, ↑Start-Up).

La barre de menu dans le bas de l'écran offre certaines fonctionnalités avancées nécessaires à quelques configurations. Par l'utilisation des touches de fonction, vous pouvez passer des options additionnelles au programme d'installation sans avoir besoin de connaître la syntaxe détaillée de ces paramètres (voir Section 2.4.2, « Utilisation d'Options de Démarrage Personnalisées » (p. 67)). Une description détaillée des touches de fonction disponibles est donnée dans Section 1.5, « L'Ecran de Démarrage » (p. 10).

2.4.2 Utilisation d'Options de Démarrage Personnalisées

L'utilisation du jeu approprié d'options de démarrage, vous facilite la procédure d'installation. De nombreux paramètres peuvent aussi être configurés plus-tard en utilisant des routines linuxrc, mais l'utilisation des options de démarrage est plus commode. Pour quelques configurations automatiques, les options de démarrage peuvent être fournies par un fichier `initrd` ou un fichier `info`

Le tableau suivant dresse la liste de tous les scénarios mentionnés dans ce chapitre avec les paramètres requis pour le démarrage et les options de démarrage associées. Ajoutez simplement chacun d'eux dans l'ordre où ils apparaissent dans ce tableau afin d'obtenir une ligne d'options de démarrage qui seront passées aux routines d'installation. Par exemple (en une seule ligne):

```
install=xxx netdevice=xxx hostip=xxx netmask=xxx vnc=xxx vncpassword=xxx
```

Remplacer toutes les valeurs `xxx` de cette ligne avec les valeurs appropriées pour votre configuration.

Chapitre 1, *Installation avec YaST* (p. 3)

Paramètre Nécessaires au Démarrage None

Boot Options (Options de Démarrage) Aucun besoin

Section 2.1.1, « Simple Installation à Distance avec VNC—Configuration Réseau Statique » (p. 38)

Paramètre Nécessaires au Démarrage

- Emplacement du Serveur d'Installation

- Périphérique réseau
- Adresse IP
- Masque de sous-réseau
- Passerelle
- Activation de VNC
- Mot de passe VNC

Boot Options (Options de Démarrage)

- `install=(nfs,http,ftp,smb):: path_to_instmedia`
- `netdevice=périphériques` (Seulement nécessaire si plusieurs périphériques réseau sont présent)
- `hostip=some_ip`
- `netmask=some_netmask`
- `gateway=ip_gateway`
- `vnc=1`
- `vncpassword=some_password`

Section 2.1.2, « Simple Installation à Distance avec VNC— Configuration Réseau Dynamique » (p. 39)

Paramètre Nécessaires au Démarrage

- Emplacement du Serveur d'Installation
- Activation de VNC
- Mot de passe VNC

Boot Options (Options de Démarrage)

- `install=(nfs,http,ftp,smb):: path_to_instmedia`
- `vnc=1`
- `vncpassword=some_password`

Section 2.1.3, « Installation à Distance via VNC—PXE Démarrage et Réveil par le LAN (Réseau Local) » (p. 41)

Paramètre Nécessaires au Démarrage

- Emplacement du Serveur d'Installation
- Emplacement du serveur TFTP
- Activation de VNC

- Mot de passe VNC

Boot Options (Options de Démarrage) Non applicable; processus dirigé à travers PXE et DHCP

Section 2.1.4, « Simple Installation à Distance via SSH—Configuration Statique du Réseau » (p. 42)

Paramètre Nécessaires au Démarrage

- Emplacement du Serveur d'Installation
- Périphérique réseau
- Adresse IP
- Masque de sous-réseau
- Passerelle
- Activation de SSH
- Mot de passe SSH

Boot Options (Options de Démarrage)

- `install=(nfs,http,ftp,smb):: path_to_instmedia`
- `netdevice=périphériques` (Seulement nécessaire si plusieurs périphériques réseau sont présent)
- `hostip=some_ip`
- `netmask=some_netmask`
- `gateway=ip_gateway`
- `usessh=1`
- `sshpassword=some_password`

Section 2.1.5, « Simple Installation à Distance via SSH—Configuration Dynamique du Réseau » (p. 44)

Paramètre Nécessaires au Démarrage

- Emplacement du Serveur d'Installation
- Activation de SSH
- Mot de passe SSH

Boot Options (Options de Démarrage)

- `install=(nfs,http,ftp,smb):: path_to_instmedia`
- `usessh=1`
- `sshpassword=some_password`

Section 2.1.6, « Installation à distance via SSH—démarrage PXE et réveil par le LAN » (p. 45)

- Emplacement du Serveur d'Installation
- Emplacement du serveur TFTP
- Activation de SSH
- Mot de passe SSH

Boot Options (Options de Démarrage) Non applicable; processus dirigé à travers PXE et DHCP

ASTUCE : Plus d'informations sur les options linuxrc de démarrage

Find more information about the linuxrc boot options used for booting a Linux system at <http://en.opensuse.org/SDB:Linuxrc>.

2.5 Surveillance du Processus d'Installation

Il existent plusieurs options pour surveiller à distance le processus d'installation. Si l'option adéquate a été spécifiée pendant le démarrage de l'installation, SSH ou VNC peuvent être utilisés pour contrôler l'installation et la configuration du système depuis une station de travail distante.

2.5.1 Installation par VNC

En utilisant un logiciel de visualisation VNC; vous pouvez contrôler à distance l'installation d' openSUSE depuis tout système d'exploitation. Cette section présente l'installation utilisant un visionneur VNC ou un navigateur Web.

2.5.1.1 Préparation pour une Installation avec VNC.

Tout ce que vous avez besoin de faire sur la cible d'installation pour préparer une installation par VNC et de fournir les options appropriées au moment du démarrage initial de l'installation (voir Section 2.4.2, « Utilisation d'Options de Démarrage Personnalisées » (p. 67)). La cible d'installation démarre dans un environnement texte et attend qu'un client VNC se connecte au programme d'installation.

Le programme d'installation annonce l'adresse IP et le numéro d'affichage nécessaire pour une connexion à l'installation. Si vous avez physiquement accès à la machine cible, cette information est fournie juste après que le système est démarré pour l'installation. Entrez ces données quand le client VNC vous invite à le faire et fournissez votre mot de passe VNC.

Comme la cible d'installation s'annonce via OpenSLP, vous pouvez récupérer les informations d'adressage de la cible d'installation via un navigateur SLP sans avoir besoin d'aucun contact physique avec celle-ci, à condition que la configuration de votre réseau et que toutes les machines en faisant partie supportent OpenSLP:

- 1 Démarrer le navigateur web et explorateur de dossier Konqueror de KDE.
- 2 Entrez `service://yast.installation.suse` dans la barre d'adresse. Le système cible apparaît alors en icône dans la page d'affichage de Konqueror. En cliquant sur cet icône le visionneur VNC se lance, permettant ainsi, d'effectuer l'installation. Vous pouvez aussi lancer votre logiciel de visionnage VNC avec l'adresse fournie précédemment et ajouter : 1 à la fin de celle-ci pour afficher l'exécution de l'installation.

2.5.1.2 Connexion au programme d'installation.

Généralement, il y a deux façons de se connecter à un serveur VNC (la cible d'installation en l'occurrence). Vous pouvez soit démarrer une application de visionnage VNC (VNC Viewer) depuis tout système d'exploitation ou vous connectez depuis un navigateur Web avec Java activé.

En utilisant VNC vous pouvez contrôler l'installation d'un système Linux depuis tout autre système d'exploitation, tels que d'autres distributions Linux, des système Windows ou Mac OS.

On a Linux machine, make sure that the package `tightvnc` is installed. On a Windows machine, install the Windows port of this application, which can be obtained at the TightVNC home page (<http://www.tightvnc.com/download.html>).

Pour se connecter au programme d'installation exécuté sur la machine cible, procédez comme suit :

- 1 Démarrez le Viewer VNC.

- 2 Entrez l'adresse IP et le numéro d'affichage de la cible d'installation tel qu'il est fourni par le navigateur SLP ou le programme d'installation lui-même :

ip_address:display_number

Une fenêtre s'ouvre sur votre bureau affichant l'écran YaST comme dans une installation locale normale.

En utilisant le navigateur Web pour se connecter au programme d'installation vous vous rendrez indépendant de tout programme VNC et du système d'exploitation sous-jacent. Du moment que le navigateur (Firefox, Internet Explorer, Konqueror, Opera, etc.) a le support Java d'activé, vous pouvez l'utiliser afin d'effectuer l'installation de votre système Linux.

Pour effectuer l'installation de VNC, procédez comme suit :

- 1 Lancez votre navigateur.
- 2 Entrez ce qui suit dans la barre d'adresse :
http://ip_address_of_target:5801
- 3 Entrez le mot de passe VNC quand vous êtes invité à le faire. Le fenêtre du navigateur affiche alors l'écran YaST comme dans une installation normale en local.

2.5.2 Installation SSH

En utilisant SSH, vous pouvez contrôler à distance l'installation de votre machine Linux depuis tout logiciel client SSH.

2.5.2.1 Préparation pour une Installation SSH

Mise à part l'installation des paquets logiciels appropriés (OpenSSH pour Linux ou PuTTY pour Windows), vous avez seulement besoin de passer les bonnes options de démarrage pour activer l'installation SSH. Voir Section 2.4.2, « Utilisation d'Options de Démarrage Personnalisées » (p. 67) pour les détails. OpenSSH est installé par défaut sur tout les systèmes basés sur SUSE Linux.

2.5.2.2 Connexion au programme d'installation.

- 1 Récupérez les adresses IP de la cible d'installation. Si vous avez un accès physique au système cible, prenez juste l'adresse IP que la routine d'installation vous fournit

en console juste après le démarrage initial. Sinon prenez l'adresse IP qui aura été assignée à cet hôte particulier dans la configuration du serveur DHCP (si celui-ci existe).

- 2** entrez les lignes suivantes dans la ligne de commande :

```
ssh -X root@  
ip_address_of_target
```

Remplacer *adresse_cible* avec la véritable adresse de la cible.

- 3** Quand vous êtes invité à entrer un nom d'utilisateur, entrez `root`.
- 4** Quand vous l'êtes à entrez un mot de passe, entrez le mot de passe qui aura été configuré grâce aux options SSH de démarrage. Après vous être authentifié avec succès, une invite de commande pour la cible d'installation apparaît.
- 5** Entrez `yast` pour lancer le programme d'installation. Une fenêtre s'ouvre montrant les écrans YaST comme décrit dans Chapitre 1, *Installation avec YaST* (p. 3).

Configuration avancée des disques

Les configurations système sophistiquées requièrent des paramétrages de disque spécifiques. Toutes les tâches de partitionnement usuelles peuvent être effectuées avec YaST. Pour obtenir des noms de périphériques de type bloc persistants, utilisez les périphériques blocs suivants `/dev/disk/by-id` ou `/dev/disk/by-uuid`. LVM (Logical Volume Management) est un schéma de partitionnement des disques conçu pour être plus flexible que le partitionnement physique utilisé dans les configurations standards. Sa fonctionnalité de snapshot permet de créer facilement des sauvegardes de données. La technologie RAID (Redundant Array of Independent Disks) offre une plus grande intégrité des données ainsi que des performances et une tolérance aux pannes accrue. openSUSE supporte aussi les E/S multichemins, et il y a aussi la possibilité d'utiliser iSCSI en tant que disque réseau.

3.1 Utilisation du partitionneur YaST

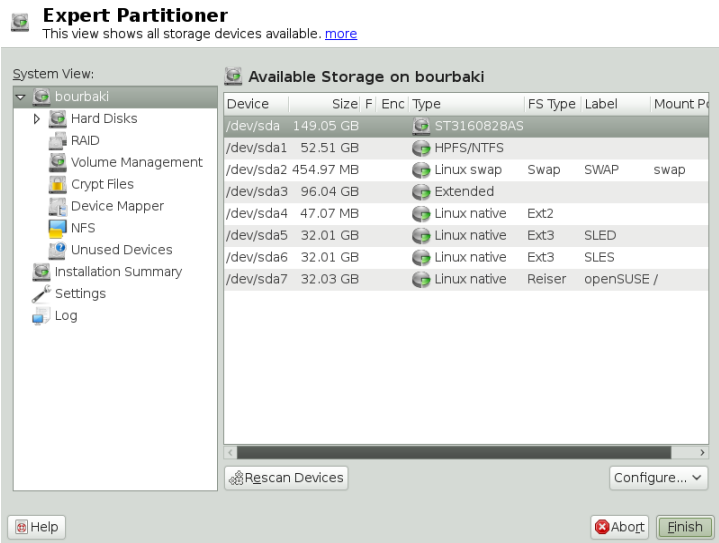
With the expert partitioner, shown in Figure 3.1, « Partitionneur YaST » (p. 76), manually modify the partitioning of one or several hard disks. You can add, delete, resize, and edit partitions, as well as access the soft RAID, and LVM configuration.

AVERTISSEMENT : Repartitionner le système d'exécution

Although it is possible to repartition your system while it is running, the risk of making a mistake that causes the data loss is very high. Try to avoid reparti-

tioning your installed system and always do a complete backup of your data before attempting to do so.

Figure 3.1 *Partitionneur YaST*



Toutes les partitions existantes ou suggérée sur tous les disques connectés sont affichées dans la liste des *Stokages Disponibles* dans la fenêtre de dialogue du *Partitionneur Expert YaST*. Les disques durs entiers sont listés comme des périphériques non numérotés, de cette façon `/dev/sda`. Les partitions sont listées entant que parties de ces périphériques, de cette façon `/dev/sda1`. La taille, le type, le status de cryptage, le système de fichiers et le point de montage des disques durs et leurs partitions sont aussi affichés. Le point de montage décrit l'endroit de l'apparition de la partition dans l'arborescence de fichier Linux.

Several functional views are available on the lefthand *System View*. Use these views to gather information about existing storage configurations, or to configure functions like RAID, Volume Management, Crypt Files, or view filesystems with additional features, such as BTRFS, NFS, or TMPFS.

If you run the expert dialog during installation, any free hard disk space is also listed and automatically selected. To provide more disk space to openSUSE®, free the needed space starting from the bottom toward the top of the list (starting from the last partition of a hard disk toward the first).

3.1.1 Types de partition

Tous les disques durs ont une table de partition avec des emplacements pour quatre entrées. Toute entrée dans la table de partition correspond à une partition primaire ou une étendue. Une partition étendue seulement, est autorisée cependant

Une partition primaire est composée simplement d'une plage continue de cylindres (espace physique du disque) assigné à un système d'exploitation spécifique. Avec des partitions primaires vous seriez limité à quatre partitions par disque, car davantage ne rentrerait pas dans la partition. C'est pourquoi les partitions étendues sont utilisées. Les partitions étendues ont elles aussi une plage continue de cylindres mais ceux-ci peuvent être divisés en *partitions logiques*. Les partitions logiques ne requièrent pas d'entrer dans la table de partition. En d'autres mots, une partition étendue est un conteneur pour des partitions logiques.

If you need more than four partitions, create an extended partition as the fourth partition (or earlier). This extended partition should occupy the entire remaining free cylinder range. Then create multiple logical partitions within the extended partition. The maximum number of logical partitions is 63, independent of the disk type. It does not matter which types of partitions are used for Linux. Primary and logical partitions both function normally.

3.1.2 Créer une partition.

Pour créer une partition à partir de zéro sélectionner *Disques Durs*, puis un disque ayant de l'espace libre. Les modifications effectives peuvent être faites dans l'onglet *Partitions* :

- 1 Select *Add* and specify the partition type (primary or extended). Create up to four primary partitions or up to three primary partitions and one extended partition. Within the extended partition, create several logical partitions (see Section 3.1.1, « Types de partition » (p. 77)).
- 2 Specify the size of the new partition. You can either choose to occupy all the free unpartitioned space, or enter a custom size.
- 3 Sélectionnez le système de fichiers à utiliser et le point de montage. YaST suggère un point de montage pour chaque partition créée. Pour utiliser une méthode de montage différente, par exemple, sélectionnez *Options Fstab*.

- 4 Spécifiez les options de système de fichiers additionnelles selon les spécificité de votre système. Cela est nécessaire, par exemple, si vous avez besoin d'un nom de périphérique persistant. Pour des détails sur les options disponibles, référez-vous à la Section 3.1.3, « Editer une partition » (p. 80).
- 5 Cliquez sur *Finish* pour appliquer votre configuration de partition et quitter le module de partitionnement.

Si vous créez la partition pendant l'installation, vous êtes ramené sur la vue d'ensemble de l'installation.

3.1.2.1 Btrfs Partitioning

If you want to use Btrfs and *Storage Administration Guide*, chapter *Overview of File Systems in Linux* for more information on Btrfs) as your default filesystem for a newly installed system, click *Partitioning* on the *Installation Settings* screen, and check *Use Btrfs as Default Filesystem*. The installation system then suggests creating the `/boot` partition formatted with Ext3 filesystem, and the root `/` partition formatted with Btrfs holding a default set of subvolumes, which you can modify with the *Expert Partitioner* tool later.

The root filesystem is the default subvolume and it is not listed in the list of created subvolumes. As a default Btrfs subvolume, it can be mounted as a normal filesystem.

It is possible to create snapshots of Btrfs subvolumes - either manually, or automatically based on system events. For example when making changes to the filesystem, `zypper` invokes the `snapper` command to create snapshots before and after the change. This is useful if you are not satisfied with the change `zypper` made and want to restore the previous state. As `snapper` invoked by `zypper` snapshots the *root* filesystem by default, it is reasonable to exclude specific directories from being snapshot, depending on the nature of data they hold. And that is why YaST suggests creating the following separate subvolumes.

Suggested Btrfs Subvolumes

`/tmp /var/tmp /var/run`
Directories with frequently changed content.

`/var/spool`
Contains user data, such as mails.

`/var/log`

Contains system and applications' log files which should never be rolled back.

`/var/crash`

Contains memory dumps of crashed kernels.

`/srv`

Contains data files belonging to FTP and HTTP servers.

`/opt`

Contains third party software.

ASTUCE : Size of Btrfs Partition

Because saved snapshots require more disk space, it is recommended to reserve more space for Btrfs partition than for a partition not capable of snapshotting (such as Ext3). Recommended size for a root Btrfs partition with suggested subvolumes is 20GB.

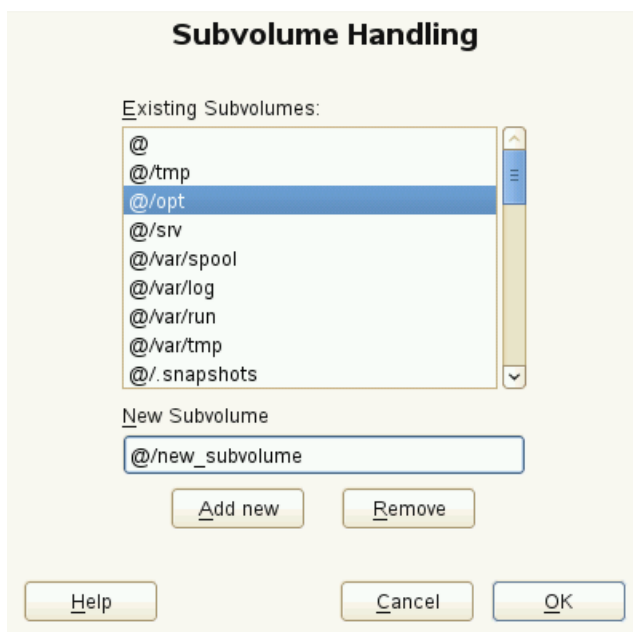
Managing Btrfs Subvolumes using YaST

Subvolumes of a Btrfs partition can be now managed with the YaST *Expert partitioner* module. You can add new or remove existing subvolumes.

Procédure 3.1 *Btrfs Subvolumes with YaST*

- 1** Start the YaST *Expert Partitioner* with *System > Partitioner*.
- 2** Choose *BTRFS* in the left *System View* pane.
- 3** Select the Btrfs partition whose subvolumes you need to manage and click *Edit*.
- 4** Click *Subvolume Handling*. You can see a list off all existing subvolumes of the selected Btrfs partition. You can notice a number of `@/.snapshots/xyz/snapshot` entries — each of these subvolumes belongs to one existing snapshot.
- 5** Depending on whether you want to add or remove subvolumes, do the following:
 - 5a** To remove a subvolume, select it from the list of *Existing Subvolumes* and click *Remove*.
 - 5b** To add a new subvolume, enter its name to the *New Subvolume* text field and click *Add new*.

Figure 3.2 *Btrfs Subvolumes in YaST Partitioner*



6 Confirm with *OK* and *Finish*.

7 Leave the partitioner with *Finish*.

3.1.3 Editer une partition

Quand vous créez une nouvelle partition ou modifiez une existante, vous pouvez paramétrer un certain nombre de paramètres. Pour une nouvelle partition, les paramètres par défaut configuré avec YaST sont habituellement suffisant et ne requièrent aucune modification. Pour éditer la configuration de votre partition manuellement, procédez comme suit :

1 Sélectionnez la partition.

2 Cliquez *Editer* pour éditer la partition et paramétrez :

ID du système de fichiers

Even if you do not want to format the partition at this stage, assign it a file system ID to ensure that the partition is registered correctly. Typical values are *Linux*, *Linux swap*, *Linux LVM*, and *Linux RAID*.

Système de fichiers

To change the partition file system, click *Format Partition* and select file system type in the *File System* list.

openSUSE supports several types of filesystems. Btrfs is the Linux filesystem of choice because of its advanced features. It supports copy-on-write functionality, creating snapshots, multi-device spanning, subvolumes, and other useful techniques. ReiserFS, JFS, XFS, and Ext3 are journaling file systems. These file systems are able to restore the system very quickly after a system crash, utilizing write processes logged during the operation. Ext2 is not a journaling file system, but it is adequate for smaller partitions because it does not require much disk space for management.

Swap is a special format that allows the partition to be used as a virtual memory. Create a swap partition of at least 256 MB. However, if you use up your swap space, consider adding more memory to your system instead of adding more swap space.

AVERTISSEMENT : Changement du système de fichiers

Le changement du système de fichiers et le reformatage des partitions efface toutes les données de la partition irréversiblement.

Périphériques cryptés

Si vous activez le cryptage, toutes les données sont écrites cryptées sur le disque dur. Ceci améliore la sécurité des données sensible, mais réduit la vitesse du système, car les données cryptées prennent plus de temps pour se lire et s'écrire. Plus d'informations sur le cryptage des systèmes de fichiers sont fournies au Chapitre 10, *Encrypting Partitions and Files* (↑Security Guide).

Point de montage.

Spécifiez le répertoire où la partition doit être montée dans l'arborescence du système de fichiers. Sélectionnez depuis YaST les suggestions ou entrez un autre nom.

Options Fstab

Vous pouvez spécifier plusieurs paramètres contenus dans le fichier global d'administration du système de fichiers (`/etc/fstab`). Les réglages par défaut devraient suffirent pour la plupart des configurations. Vous pouvez, par exemple, changer le nom du périphérique en étiquette de volume. Dans une étiquette de volume, utilisez tous les caractères exceptés `/` et espace.

Pour obtenir des noms de périphériques persistents, utilisez l'option de montage *ID du périphérique*, *UUID* ou *LABEL*. Dans `productname`;, les noms de périphériques persistants sont activés par défaut.

If you prefer to mount the partition by its label, you need to define one in the *Volume label* text entry. For example, you could use the partition label `HOME` for a partition intended to mount to `/home`.

Si vous souhaitez utiliser des quotas sur le système de fichiers, utilisez l'option de montage *Activer la gestion des quotas*. Ceci doit être fait avant la définition des quotas selon les utilisateurs dans *Gestion des utilisateurs* du module YaST. Pour davantage d'informations sur comment configurer les quotas utilisateurs, référez-vous à la section « Managing Quotas » (Chapitre 10, *Managing Users with YaST*, ↑Start-Up).

3 Sélectionnez *Finish* pour sauvegarder les changements.

REMARQUE : Redimensionner le système de fichiers

Pour redimensionner un système de fichiers existant, sélectionnez la partition et utilisez *Redimensionner*. Notez qu'il n'est pas possible de redimensionner les partitions quand celles-ci sont montées. Pour redimensionner les partitions, démonter la partition concernée avant d'exécuter le partitionneur.

3.1.4 Options Expertes

Après avoir sélectionné un disque dur (tel que *sda*) dans le panneau *Vue Système*, vous pouvez accéder le menu *Expert...* dans la partie en bas à droite de la fenêtre *Partitionneur Expert*. Le menu contient les commandes suivantes :

Créer une nouvelle table de partition

Cette option vous aide à créer une nouvelle table de partition sur le périphérique sélectionné.

AVERTISSEMENT : Création d'une nouvelle table de partition

La création d'une nouvelle table de partition sur un périphérique enlève irréversiblement toutes les partitions et leurs données de ce périphérique.

Cloner ce disque

This option helps you clone the device partition layout (but not the data) to other available disk devices.

3.1.5 Options avancés

Après que vous ayez sélectionné le nom d'hôte de votre ordinateur (le niveau le plus haut de l'arbre dans le panneau de la *Vue Système*), vous pouvez accéder au menu *Configurer...* en bas à droite dans la fenêtre du *Partitionneur Expert*. Ce menu contient les commandes suivantes :

Configurer iSCSI

Pour accéder à un périphérique SCSI sur IP en mode block, vous devez tout d'abord configurer iSCSI. Cela a pour résultat l'addition des périphériques disponibles à la liste principale des partitions.

Configurer le multipath.

Choisir cette option vous aidera à configurer l'amélioration du multipath (ou multitrajets) dans les périphériques de masse supportés.

3.1.6 Plus de conseils de partitionnement

La section suivante inclut quelques trucs et astuces sur le partitionnement qui pourraient vous aider à prendre la bonne décision pour le paramétrage de votre système.

ASTUCE : Nombres de cylindres.

Notez, que les différents outils de partitionnement peuvent commencer le compte des cylindres à partir de 0 ou de 1. Lors du calcul du nombre de cylindres, vous devez toujours utiliser la différence entre le premier et le dernier numéro de cylindre et ajouter un.

3.1.6.1 Utilisation du swap

Le swap est utilisé pour étendre la mémoire physique disponible. Il est alors possible d'utiliser plus de mémoire que la mémoire physique disponible. Le système de gestion de la mémoire des noyau avant 2.4.10 nécessit  le swap en tant que mesure de s curit . Ces limitations n'existent plus dor navant.

Linux utilise une page appel e « Least Recently Used » (LRU moins r cemment utilis e) pour s lectionner les pages qui doivent  tre transf r e de la RAM vers le disque. Par cons quent, les applications active du syst me dispose de plus de m moire et la mise en cache se fait en douceur.

Si une application essaie d'allouer le maximum de la m moire autoris e, des problemes de swap peuvent survenir. Il existent trois sc narios possibles :

Le syst me n'a plus de swap

L'application obtient le maximum de la m moire autoris e. Tous les caches sont lib r s, et toutes les autres applications sont ralenties. Apr s quelques minutes, le m canisme de gestion des d passement m moire du noyau se d clenche et tue le processus fautif.

Syst me avec un taille de swap moyenne (128 MB–512 MB)

Au d but le syst me est lent comme un syst me d pourvu de swap. Apr s que toutes la m moire physique a  t  allou e, l'espace de swap est utilis  au mieux. A ce point, le syst me devient tr s lent et il devient impossible d'ex cuter une commande   distance. Selon la vitesse des disques durs qui accueillent l'espace de swap, le syst me reste dans cet  tat pendant 10   15 mn jusqu'  ce que le m canisme de gestion du d passement m moire resolve le probl me. Notes que vous aurez besoin d'une certaines quantit  de swap si l'ordinateur a besoin d'effectuer une « hibernation ». Dans ce cas, la taille du swap doit  tre assez large pour contenir les donn es utiles de la m moire. (512 MB–1GB).

Syst me avec beaucoup de swap (plusieurs Go)

Il vaut mieux ne pas avoir d'application qui soit hors de contr le et effectue de swapping excessif dans ce cas. Si vous utilisez de telles applications, le syst me aura besoin de plusieurs heures pour restaurer. Durant le processus, il est probable que les autres processus g n rent des erreurs et des d passement du d lai d'attente, laissant le syst me dans un  tat ind fini, m me apr s avoir tu  le processus fautif. Dans ce cas, faites un reboot sauvage et essayez de l'ex cuter de nouveau. Beaucoup de swap n'est seulement n cessaire si vous avez une applica-

tion qui s'appuie sur cette fonctionnalité. De telles applications (comme les bases de données ou les programmes de graphisme) ont souvent une option pour utiliser un espace du disque pour leurs besoins. Il est conseillé d'utiliser cette option au lieu d'utiliser trop d'espace swap.

Si votre système n'est pas hors de contrôle, mais nécessite plus de swap ou bout d'un certain temps, il est possible d'étendre l'espace de swap en ligne. Si vous avez préparé une partition pour l'espace de swap, simplement ajoutez cette partition avec YaST. Les fichiers de swap sont généralement moins rapides que les partitions, mais cette différence est négligeable comparativement à leurs lenteurs mutuelles par rapport à la mémoire physique.

Procédure 3.2 *Ajouter un fichier de swap manuellement.*

Pour ajouter un fichier de swap manuellement au système en cours d'exécution, procédez comme suit :

- 1** Créez un fichier vide dans votre système. Par exemple, si vous voulez ajouter un fichier swap avec 128 Mo de swap à `/var/lib/swap/swapfile`, utilisez les commandes :

```
mkdir -p /var/lib/swap
dd if=/dev/zero of=/var/lib/swap/swapfile bs=1M count=128
```

- 2** Initialisez ce fichier de swap avec la commande

```
mkswap /var/lib/swap/swapfile
```

- 3** Activez le swap avec cette commande

```
swapon /var/lib/swap/swapfile
```

Pour désactiver ce fichier de swap, utilisez :

```
swapoff /var/lib/swap/swapfile
```

- 4** Vérifiez les espaces de swap courant disponible avec la commande :

```
cat /proc/swaps
```

Notez que à ce point, il s'agit seulement d'un espace de swap temporaire. Après le prochain reboot, il ne sera plus utilisé.

- 5** Pour activer ce fichier de swap en permanence, ajoutez les lignes ci-dessous à `/etc/fstab`:

```
/var/lib/swap/swapfile swap swap defaults 0 0
```

3.1.7 Partitionnement et LVM

From the *Expert partitioner*, access the LVM configuration by clicking the *Volume Management* item in the *System View* pane. However, if a working LVM configuration already exists on your system, it is automatically activated upon entering the initial LVM configuration of a session. In this case, all disks containing a partition (belonging to an activated volume group) cannot be repartitioned. The Linux kernel cannot reread the modified partition table of a hard disk when any partition on this disk is in use. If you already have a working LVM configuration on your system, physical repartitioning should not be necessary. Instead, change the configuration of the logical volumes.

Au début des volumes physiques (PVs), l'information est écrite sur la partition. Pour le réutiliser comme une partition à d'autres fins que du LVM, il est conseillé d'effacer le début de ce volume. Par exemple, dans le VG `system` et le PV `/dev/sda2`, faire ceci avec la commande `dd if=/dev/zero of=/dev/sda2 bs=512 count=1`.

AVERTISSEMENT : Système de fichiers pour le démarrage

Le système de fichiers utilisé pour le démarrage (le système de fichiers racine ou `/boot`) ne doivent pas être stocké sur un volume logique LVM. Stockez-le plutôt sur une partition normale.

3.2 Configuration LVM

Cette section décrit brièvement les principes de la gestion de volumes logiques (LVM Logical Volume Manager) et ces fonctionnalités polyvalentes. A la Section 3.2.2, « Configuration LVM avec YaST » (p. 88), vous apprendrez comment configurer LVM avec YaST.

AVERTISSEMENT

L'utilisation de LVM est parfois associée à un risque plus élevé de perte de données. Ces risques incluent aussi les crash d'applications, dysfonctionnement de l'alimentation et les commandes défectueuses. Sauvegardez vos

données avant d'implémenter LVM ou reconfigurer les volumes. Ne travaillez jamais sans une sauvegarde de vos données.

3.2.1 Le Gestionnaire de Volume Logique.

Le LVM permet une distribution flexible de l'espace disque dur avec plusieurs systèmes de fichiers. Il a été développé car quelquefois le besoins de changer la segmentation de l'espace disque survient juste après que le partitionnement initial a été fait. Comme il est difficile de modifier les partitions sur un système en cours d'exécution, LVM fournit un groupe virtuel (volume group, VG) d'espace mémoire sur laquelle les volumes logiques (LV) peuvent être créés. Le système d'exploitation accède a ces volumes logiques au lieu des partitions physiques. Les groupes de volumes peuvent occuper plus d'un disque, de telles sortes que plusieurs disque ou parties de disque peuvent constituer un simple groupe de volume. Ainsi, LVM fournit une sorte d'abstraction du disque physique qui permet à sa segmentation d'être changé plus facilement et plus sûrement qu'une repartition physique. Des informations générales concernant le partitionnement physique peuvent être trouvées à la Section 3.1.1, « Types de partition » (p. 77) et la Section 3.1, « Utilisation du partitionneur YaST » (p. 75).

Figure 3.3 *Partitionnement Physique et LVM*

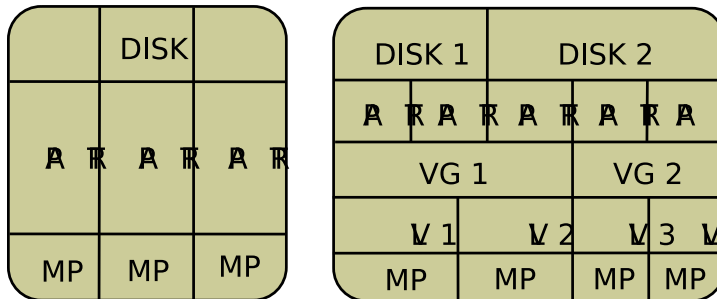


Figure 3.3, « Partitionnement Physique et LVM » (p. 87) compares physical partitioning (left) with LVM segmentation (right). On the left side, one single disk has been divided into three physical partitions (PART), each with a mount point (MP) assigned so that the operating system can gain access. On the right side, two disks have been divided into two and three physical partitions each. Two LVM volume groups (VG 1 and VG 2) have been defined. VG 1 contains two partitions from DISK 1 and one from DISK 2. VG 2 contains the remaining two partitions from DISK 2. In LVM, the physical disk partitions that are incorporated in a volume

group are called physical volumes (PVs). Within the volume groups, four LVs (LV 1 through LV 4) have been defined. They can be used by the operating system via the associated mount points. The border between different LVs do not need to be aligned with any partition border. See the border between LV 1 and LV 2 in this example.

Fonctionnalités LVM :

- Plusieurs disques durs ou partitions peuvent être combinés dans un grand volume logique.
- Pouvu que la configuration soit adaptée, un Volume logique (tel que `/usr`) peut être agrandi si l'espace disque est suffisant.
- Avec LVM, il est possible d'ajouter des disques durs ou des volumes logiques dans un système en cours d'exécution. Toutefois, cela requiert du matériel remplaçable à chaud.
- Il est possible d'activer un "striping mode" qui distribue le flux des données d'un volume logique sur plusieurs volumes physiques. Si ces volumes physiques résident sur des disques différents, les performances de lecture et d'écriture sont améliorées, de la même façon qu'avec un RAID 0.
- La fonctionnalité snapshot permet des sauvegardes cohérentes (spécialement pour les serveurs) du système en cours d'exécution.

With these features, LVM is ready for heavily used home PCs or small servers. LVM is well-suited for the user with a growing data stock (as in the case of databases, music archives, or user directories). This would allow file systems that are larger than the physical hard disk. Another advantage of LVM is that up to 256 LVs can be added. However, working with LVM is different from working with conventional partitions. Instructions and further information about configuring LVM is available in the official LVM HOWTO at <http://tldp.org/HOWTO/LVM-HOWTO/>.

Depuis la version de noyau 2.6, LVM version 2 est disponible. Celle-ci ne requiert aucun patch pour le noyau. Il permet l'utilisation du mappeur de p 2. C'est pourquoi, quand nous parlerons de LVM dans cette section, nous nous r

3.2.2 Configuration LVM avec YaST

La configuration de LVM avec YaST peut être atteinte depuis le partitionneur Expert YaST (see Section 3.1, « Utilisation du partitionneur YaST » (p. 75)) dans le

module *Gestionnaire de Volume* du panneau *Vue système*. Le partitionneur expert vous permet d'éditer et effacer les partitions existantes, mais aussi de créer les nouvelles dont vous avez besoin pour utiliser LVM. La première tâche est de créer les volumes physiques qui mettent un espace à disposition des groupes de volumes :

- 1 Choisir un disque dur depuis *Disques Durs*.
- 2 Aller dans l'onglet *Partitions*.
- 3 Cliquez *Ajouter* et entrez la taille désirée pour le volume physique sur ce disque.
- 4 Utilisez *Ne pas formater la Partition* et changer le *File System ID* en *0x8E Linux LVM*. Ne montez pas cette partition.
- 5 Répétez cet procédure jusqu'à ce que vous ayez défini tous les volumes physique que vous souhaitez sur les disques disponibles.

3.2.2.1 Création des groupes de volumes

Si aucun groupe de volume n'existe sur votre système, vous devez en ajouter un (voir Figure 3.4, « Création d'un groupe de volume » (p. 90)). Il est possible de créer des groupes additionnelles en cliquant sur *Gestionnaire de Volume* dans le panneau *Vue Système*, puis sur *Ajouter un Groupe de Volume*. Un simple groupe de volume est habituellement suffisant.

- 1 Entrer un nom pour le groupe de volume, par exemple, `system`.²
- 2 Sélectionner la *Taille Physique Etendue* désirée. Cette valeur définit la taille d'un bloc physique dans le groupe de volume. Tout l'espace disque dans un groupe de volume est manipulé sous la forme de blocs de cette taille.
- 3 Ajoutez les volumes physiques préparés au groupe de volume en sélectionnant le pé et en cliquant sur *Ajouter*. `SCtrl` pendant le s
- 4 Sélectionnez *Finir* pour rendre disponible le groupe de volume pour les étapes de configuration suivantes.

Figure 3.4 *Création d'un groupe de volume*

Add Volume Group
Enter the name and physical extent size of the new volume group. [more](#)

Volume Group Name:

Physical Extent Size:

Available Physical Volumes:

Device	Size	Enc	Type
/dev/sda1	101.94 MB		Linux native
/dev/sda2	62.75 MB		Linux native
/dev/sda6	20.00 GB		Linux native
/dev/sda7	30.01 GB		Linux native

Selected Physical Volumes:

Device	Size	Enc	Type
No entries.			

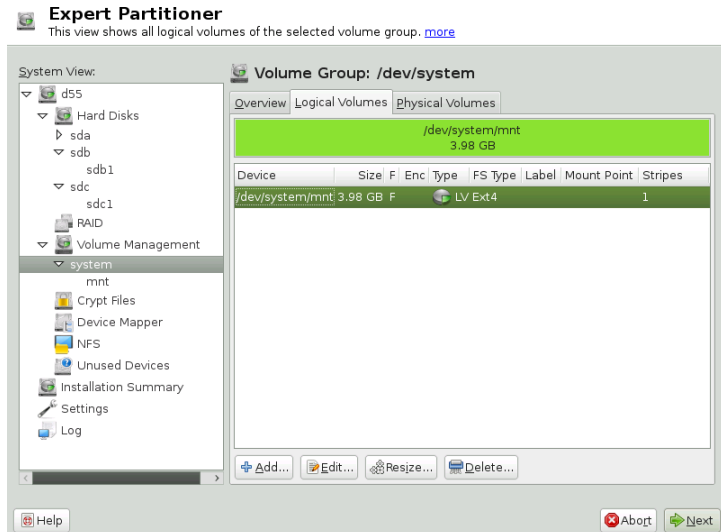
Total size: 50.17 GB Resulting size: 0.00 B

If you have multiple volume groups defined and want to add or remove PVs, select the volume group in the *Volume Management* list and click *Resize*. In the following window, you can add or remove PVs to the selected volume group.

3.2.2.2 Configuration des volumes logiques

Après avoir rempli le groupe de volume avec des volumes physiques, définissez le volume logique que le système d'exploitation doit utiliser dans la fenêtre suivante. Choisissez le groupe de volume courant et rendez-vous dans l'onglet `guimenu>Volumes Logiques`

Figure 3.5 Gestionnaire de Volume Logique



Cliquez *Ajouter* et passez le pop-up de l'assistant qui s'ouvre :

1. Entrez le nom du volume logique. Pour une partition qui peut être montée sur / home, Un nom explicite tel que HOME peut être utilisé.
2. Sélectionnez la taille et le nombre de bandes du volume logique. Si vous avez seulement un volume physique, la sélection de plusieurs bandes est inutile.
3. Choisissez le système de fichier à utiliser sur le volume logique ainsi que le point de montage.

En utilisant les bandes il est possible de distribuer le flux de données dans le volume logique parmi plusieurs volume physique (entrelacement). Cependant, l'entrelacement d'un volume ne peut seulement fonctionner sur des volumes physiques différents, chacun fournissant au moins la quantité de l'espace du volume. Le nombre maximum de bandes est égale au nombre de volumes physiques, où bande "1" signifie "no striping" (ndt : pas d'entrelacement). L'entrelacement n'a de sens qu'avec plusieurs disque durs différents, autrement les performances seront diminuées.

AVERTISSEMENT : Entrelacement

YaST ne peut pas à ce stade vérifier la validité de vos entrées concernant l'entrelacement. Toute erreur ici ne sera apparente une fois seulement que le LVM sera implanter sur le disque.

Si vous avez déjà configuré LVM sur votre système, les volumes logiques peuvent aussi être utilisés. Avant de continuer, assignez les point de montages appropriés pour ces volumes logiques. Avec *Finir* retournez sur le Partitionneur expert de YaST et finissez-y vos travaux.

3.3 Soft RAID Configuration

The purpose of RAID (redundant array of independent disks) is to combine several hard disk partitions into one large *virtual* hard disk to optimize performance and/or data security. Most RAID controllers use the SCSI protocol because it can address a larger number of hard disks in a more effective way than the IDE protocol. It is also more suitable for the parallel command processing. There are some RAID controllers that support IDE or SATA hard disks. Soft RAID provides the advantages of RAID systems without the additional cost of hardware RAID controllers. However, this requires some CPU time and has memory requirements that make it unsuitable for high performance computers.

With openSUSE®, you can combine several hard disks into one soft RAID system. RAID implies several strategies for combining several hard disks in a RAID system, each with different goals, advantages, and characteristics. These variations are commonly known as *RAID levels*.

Common RAID levels are:

RAID 0

This level improves the performance of your data access by spreading out blocks of each file across multiple disk drives. Actually, this is not really a RAID, because it does not provide data backup, but the name *RAID 0* for this type of system is commonly used. With RAID 0, two or more hard disks are pooled together. Performance is enhanced, but the RAID system is destroyed and your data lost if even one hard disk fails.

RAID 1

This level provides adequate security for your data, because the data is copied to another hard disk 1:1. This is known as *hard disk mirroring*. If one disk is destroyed, a copy of its contents is available on the other one. All disks but one could be damaged without endangering your data. However, if the damage is not detected, the damaged data can be mirrored to the undamaged disk. This could result in the same loss of data. The writing performance suffers in the copying process compared to using single disk access (10 to 20 % slower), but read access is significantly faster in comparison to any one of the normal physical hard disks. The reason is that the duplicate data can be parallel-scanned. Generally it can be said that Level 1 provides nearly twice the read transfer rate of single disks and almost the same write transfer rate as single disks.

RAID 5

RAID 5 is an optimized compromise between Level 0 and Level 1, in terms of performance and redundancy. The hard disk space equals the number of disks used minus one. The data is distributed over the hard disks as with RAID 0. *Parity blocks*, created on one of the partitions, exist for security reasons. They are linked to each other with XOR, enabling the contents to be reconstructed by the corresponding parity block in case of system failure. With RAID 5, no more than one hard disk can fail at the same time. If one hard disk fails, it must be replaced as soon as possible to avoid the risk of losing data.

RAID 6

To further increase the reliability of the RAID system, it is possible to use RAID 6. In this level, even if two disks fail, the array still can be reconstructed. With RAID 6, at least 4 hard disks are needed to run the array. Note that when running as software raid, this configuration needs a considerable amount of CPU time and memory.

RAID 10 (RAID 1+0)

This RAID implementation combines features of RAID 0 and RAID 1: the data are first mirrored in separate disk arrays, which are inserted into a new RAID 0; type array. In each RAID 1 sub-array, one disk can fail without any damage to the data. RAID 10 is used for database application where a huge load is expected.

Other RAID Levels

Several other RAID levels have been developed (RAID 2, RAID 3, RAID 4, RAIDn, RAID 10, RAID 0+1, RAID 30, RAID 50, etc.), some of them being proprietary implementations created by hardware vendors. These levels are not very common and therefore are not explained here.

3.3.1 Soft RAID Configuration with YaST

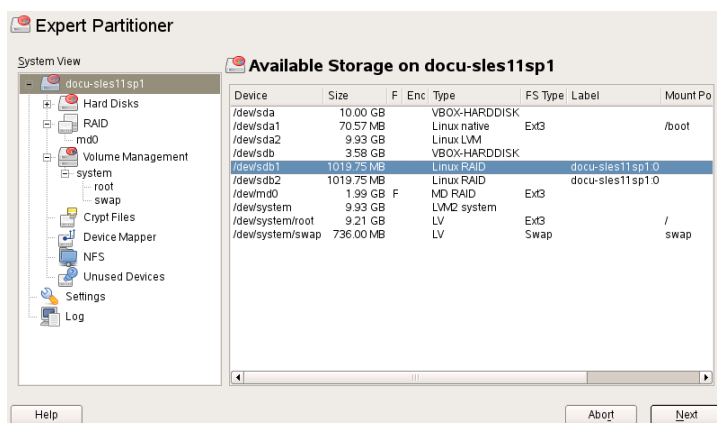
The YaST *RAID* configuration can be reached from the YaST Expert Partitioner, described in Section 3.1, « Utilisation du partitionneur YaST » (p. 75). This partitioning tool enables you to edit and delete existing partitions and create new ones to be used with soft RAID:

- 1 Select a hard disk from *Hard Disks*.
- 2 Change to the *Partitions* tab.
- 3 Click *Add* and enter the desired size of the raid partition on this disk.
- 4 Use *Do not Format the Partition* and change the *File System ID* to *0xFD Linux RAID*. Do not mount this partition.
- 5 Repeat this procedure until you have defined all the desired physical volumes on the available disks.

For RAID 0 and RAID 1, at least two partitions are needed—for RAID 1, usually exactly two and no more. If RAID 5 is used, at least three partitions are required. It is recommended to utilize partitions of the same size only. The RAID partitions should be located on different hard disks to decrease the risk of losing data if one is defective (RAID 1 and 5) and to optimize the performance of RAID 0. After creating all the partitions to use with RAID, click *RAID > Add RAID* to start the RAID configuration.

In the next dialog, choose between RAID levels 0, 1, 5, 6 and 10. Then, select all partitions with either the « Linux RAID » or « Linux native » type that should be used by the RAID system. No swap or DOS partitions are shown.

Figure 3.6 RAID Partitions



To add a previously unassigned partition to the selected RAID volume, first click the partition then *Add*. Assign all partitions reserved for RAID. Otherwise, the space on the partition remains unused. After assigning all partitions, click *Next* to select the available *RAID Options*.

In this last step, set the file system to use as well as encryption and the mount point for the RAID volume. After completing the configuration with *Finish*, see the `/dev/md0` device and others indicated with *RAID* in the expert partitioner.

3.3.2 Troubleshooting

Check the file `/proc/mdstat` to find out whether a RAID partition has been damaged. In the event of a system failure, shut down your Linux system and replace the defective hard disk with a new one partitioned the same way. Then restart your system and enter the command `mdadm /dev/mdX --add /dev/sdX`. Replace 'X' with your particular device identifiers. This integrates the hard disk automatically into the RAID system and fully reconstructs it.

Note that although you can access all data during the rebuild, you may encounter some performance issues until the RAID has been fully rebuilt.

3.3.3 For More Information

Configuration instructions and more details for soft RAID can be found in the HOWTOs at:

- `/usr/share/doc/packages/mdadm/Software-RAID.HOWTO.html`
- <http://raid.wiki.kernel.org>

Linux RAID mailing lists are available, such as <http://marc.info/?l=linux-raid>.

Partie II. Advanced Administration

Snapshots/Rollback with Snapper

Being able to do file system snapshots on Linux providing the ability to do rollbacks is a feature that was often requested in the past. Snapper, in conjunction with the `Btrfs` file system now fills that gap.

`Btrfs`, a new copy-on-write file system for Linux, supports file system snapshots (a copy of the state of a subvolume at a certain point of time) of subvolumes (one or more separately mountable file systems within each physical partition). Snapper lets you manage these snapshots. Snapper comes with a command line and a YaST interface.

By default Snapper and `Btrfs` on openSUSE are set up to serve as an « undo tool » for system changes made with YaST and `zypper`. Before and after running a YaST module or `zypper`, a snapshot is created. Snapper lets you compare the two snapshots and provides means to revert the differences between the two snapshots. The tools also provide system backups by creating hourly snapshots of the system subvolumes.

4.1 Requirements

Since `Btrfs` is the only filesystem on openSUSE supporting snapshots, it is required on all partitions or subvolumes you want to « snapshot ».

4.1.1 snapshots and Disk Space

When a snapshot is created, both the snapshot and the original point to the same blocks in the file system. So, initially a snapshot does not occupy additional disk

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

REMARQUE : Snapshot Location

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

As a result, partitions containing snapshots need to be larger than « normal » partitions. The exact amount strongly depends on the number of snapshots you keep and the amount of data modifications. As a rule of thumb you should consider using twice the size than you normally would.

ASTUCE : Freeing space / Disk Usage

In order to free space on a `Btrfs` partition containing snapshots you need to delete unneeded snapshots rather than files. Older snapshots occupy more space than recent ones.

Since the `df` does not show the correct disk usage on `Btrfs` file systems, you need to use the command `btrfs filesystem df MOUNT_POINT`. Displaying the amount of disk space a snapshot allocates is currently not supported by the `Btrfs` tools.

4.2 Using Snapper to Undo YaST and zypper Changes

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

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

Since Linux is a multitasking system, processes other than YaST or zypper may modify data in the timeframe between the pre- and the post-snapshot. If this is the case, completely reverting to the pre-snapshot will also undo these changes by other processes. In most cases this would be unwanted—therefore it is strongly recommended to closely review the changes between two snapshots before starting the rollback. If there are changes from other processes you want to keep, select which files to roll back.

IMPORTANT : Limitations

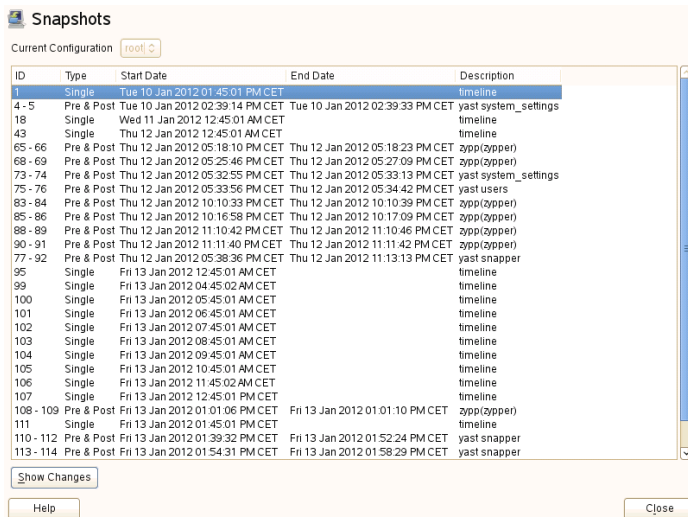
Make sure you know about Snapper's limitations before attempting to use its rollback mechanism. See Section 4.7, « Limitations » (p. 113) for details.

REMARQUE : Storage Time of Snapshots

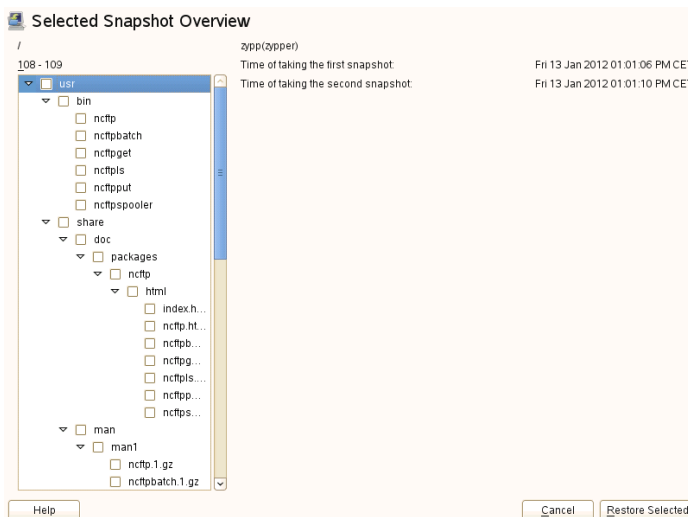
By default, the last 100 YaST and zypper snapshots are being kept. If this number is exceeded, the oldest snapshot(s) will be deleted.

Procédure 4.1 *Undoing changes using the YaST Snapper module*

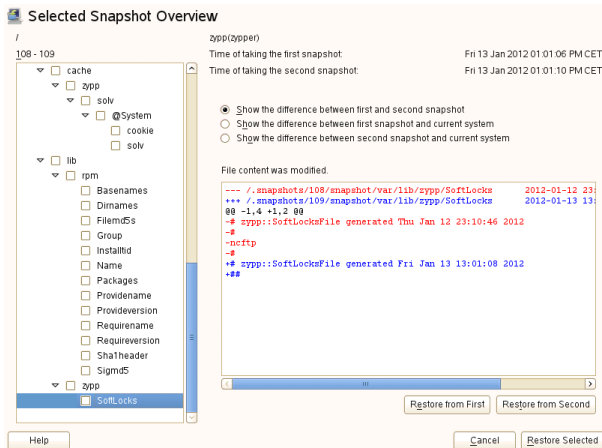
- 1 Start the *Snapper* module from the *Miscellaneous* section in YaST or by entering `yast2 snapper`.
- 2 Make sure *Current Configuration* is set to *root*. This is always the case unless you have manually added own Snapper configurations.
- 3 Choose a pair of pre- and post-snapshots from the list.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



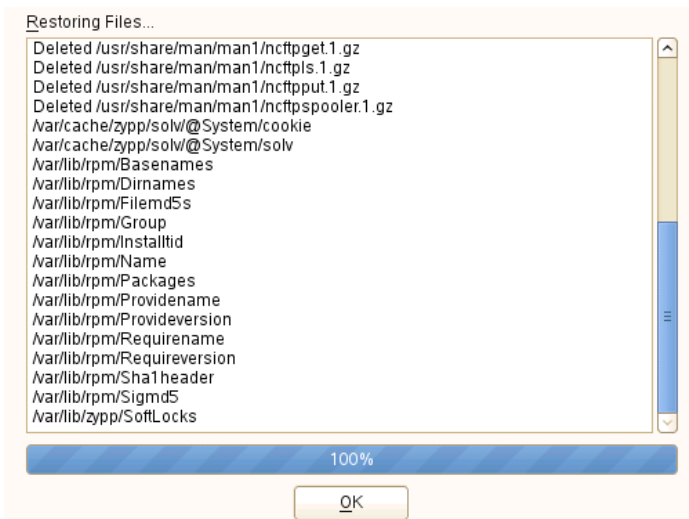
- 4 Click *Show Changes* to open the list of files that differ between the two snapshots. The following image shows a list of files that have changed after having added the user `tester`.



- 5 Review the list of files. To display a « diff » between the pre- and post-version of a file, select it from the list. The following images shows the changes to `/etc/passwd` after having added the user `tester`.



- 6 To restore a set of files, select the relevant files or directories by ticking the respective checkbox. Click *Restore Selected* and confirm the action by clicking *Yes*.



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

Procédure 4.2 Undoing changes using the snapper command

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

```
~ # snapper list -t pre-post
Pre # | Post # | Pre Date                | Post Date                | Description
-----+-----+-----+-----+-----
+
4      | 5       | Tue Jan 10 14:39:14 2012 | Tue Jan 10 14:39:33 2012 | yast system_settings
65     | 66      | Thu Jan 12 17:18:10 2012 | Thu Jan 12 17:18:23 2012 | zypp(zypper)
68     | 69      | Thu Jan 12 17:25:46 2012 | Thu Jan 12 17:27:09 2012 | zypp(zypper)
73     | 74      | Thu Jan 12 17:32:55 2012 | Thu Jan 12 17:33:13 2012 | yast system_settings
75     | 76      | Thu Jan 12 17:33:56 2012 | Thu Jan 12 17:34:42 2012 | yast users
77     | 92      | Thu Jan 12 17:38:36 2012 | Thu Jan 12 23:13:13 2012 | yast snapper
83     | 84      | Thu Jan 12 22:10:33 2012 | Thu Jan 12 22:10:39 2012 | zypp(zypper)
85     | 86      | Thu Jan 12 22:16:58 2012 | Thu Jan 12 22:17:09 2012 | zypp(zypper)
88     | 89      | Thu Jan 12 23:10:42 2012 | Thu Jan 12 23:10:46 2012 | zypp(zypper)
90     | 91      | Thu Jan 12 23:11:40 2012 | Thu Jan 12 23:11:42 2012 | zypp(zypper)
108    | 109     | Fri Jan 13 13:01:06 2012 | Fri Jan 13 13:01:10 2012 | zypp(zypper)
```

- 2 Get a list of changed files for a snapshot pair with `snapper status PRE..POST`. Files with content changes are marked with `c`, files that have been added are marked with `+` and deleted files are marked with `-`. The following example shows a snapshot pair for the installation of the package `ncftp`.

```
~ # snapper status 108..109
+... /usr/bin/ncftp
+... /usr/bin/ncftpbatch
+... /usr/bin/ncftpget
+... /usr/bin/ncftpls
[... ]
+... /usr/share/man/man1/ncftpspooler.1.gz
c... /var/cache/zypp/solv/@System/cookie
c... /var/cache/zypp/solv/@System/solv
c... /var/lib/rpm/Basenames
c... /var/lib/rpm/Dirnames
c... /var/lib/rpm/Filemd5s
c... /var/lib/rpm/Group
c... /var/lib/rpm/Installtid
c... /var/lib/rpm/Name
c... /var/lib/rpm/Packages
c... /var/lib/rpm/Providename
c... /var/lib/rpm/Provideversion
c... /var/lib/rpm/Requirename
c... /var/lib/rpm/Requireversion
c... /var/lib/rpm/Shalheader
c... /var/lib/rpm/Sigmd5
c... /var/lib/zypp/SoftLocks
```

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

```
~ # snapper diff 108..109 /var/lib/zypp/SoftLocks
--- /.snapshots/108/snapshot/var/lib/zypp/SoftLocks 2012-01-12
    23:15:22.408009164 +0100
+++ /.snapshots/109/snapshot/var/lib/zypp/SoftLocks 2012-01-13
    13:01:08.724009131 +0100
@@ -1,4 +1,2 @@
-# zypp::SoftLocksFile generated Thu Jan 12 23:10:46 2012
-#
-ncftp
-#
+# zypp::SoftLocksFile generated Fri Jan 13 13:01:08 2012
+##
```

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

```
~ # snapper -v undochange 108..109
create:0 modify:16 delete:21
undoing change...
deleting /usr/share/man/man1/ncftpspooler.1.gz
deleting /usr/share/man/man1/ncftpput.1.gz
[...]
deleting /usr/bin/ncftpls
deleting /usr/bin/ncftpget
deleting /usr/bin/ncftpbatch
deleting /usr/bin/ncftp
modifying /var/cache/zypp/solv/@System/cookie
modifying /var/cache/zypp/solv/@System/solv
modifying /var/lib/rpm/Basenames
modifying /var/lib/rpm/Dirnames
modifying /var/lib/rpm/Filemd5s
modifying /var/lib/rpm/Group
modifying /var/lib/rpm/Installtid
modifying /var/lib/rpm/Name
modifying /var/lib/rpm/Packages
modifying /var/lib/rpm/Providename
modifying /var/lib/rpm/Provideversion
modifying /var/lib/rpm/Requirename
modifying /var/lib/rpm/Requireversion
modifying /var/lib/rpm/Shalheader
modifying /var/lib/rpm/Sigmd5
modifying /var/lib/zypp/SoftLocks
undoing change done
```

4.3 Using Snapper to Restore Files from Hourly Backups

Apart from the YaST and zypper snapshots, Snapper creates hourly snapshots of the system partition (/). You can use these backup snapshots to restore files that have accidentally been deleted or modified beyond recovery. By making use of Snapper's diff feature you can also find out which modifications have been made at a certain point of time.

Hourly backup snapshots are of the type `Single` and are marked with the description `timeline`. To restore files from these snapshots proceed as described in Procédure 4.1, « Undoing changes using the YaST *Snapper* module » (p. 101) or Procédure 4.2, « Undoing changes using the `snapper` command » (p. 104)

REMARQUE : Storage Time of Snapshots

By default, the last ten snapshots of each day are kept. Furthermore ten snapshots are kept for the last ten days, months, and years. For details see Exemple 4.1, « Example timeline configuration » (p. 112).

4.4 Managing Snapshots

With Snapper you can not only create and delete snapshots, but also set and modify the metadata. All Snapper operations are carried out for a certain configuration. If you do not specify a configuration, the default one (`root`) is used. If you want to manage snapshots for your own configuration (see Section 4.6, « Creating Snapper Configurations » (p. 110) for details), you need to specify it with the global `snapper` option `-c`:

```
snapper -c myconfig list
```

For details on `snapper` refer to its manpage (`man 1 snapper`).

4.4.1 Snapshot Metadata

Each snapshot consists of the snapshot itself and some metadata. With Snapper you can set and modify some metadata. The following metadata is available for each snapshot:

- **Type:** Snapshot type, see Section 4.4.1.1, « Snapshot Types » (p. 107) for details. This data can not be changed.
- **Number:** Unique number of the snapshot. This data can not be changed.
- **Pre Number:** Specifies the number of the corresponding pre snapshot. For snapshots of type post only. This data can not be changed.
- **Description:** A description of the snapshot.
- **Userdata:** An extended description where you can specify custom data in the form or comma-separated key=value list: `reason=testing_stuff, user=tux`
- **Cleanup-Algorithm:** Cleanup-algorithm for the snapshot

4.4.1.1 Snapshot Types

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

`pre`

Snapshot of a filesystem *before* a modification. Each `pre` snapshot has got a corresponding `post` snapshot. Used e.g. for the automatic YaST/zypper snapshots.

`post`

Snapshot of a filesystem *after* a modification. Each `post` snapshot has got a corresponding `pre` snapshot. Used e.g. for the automatic YaST/zypper snapshots.

`single`

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

4.4.1.2 Cleanup-algorithms

Snapper provides three algorithms to cleanup old snapshots. The algorithms are executed in a daily cron-job. The cleanup-frequency itself is defined in the Snapper configuration for the partition or subvolume (see Section 4.6.1, « Adjusting the Config File » (p. 111) for details).

number

Deletes old snapshots when a certain snapshot count is reached.

timeline

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

empty-pre-post

Deletes pre/post snapshot pairs with empty diffs.

4.4.2 Creating Snapshots

Creating a snapshot is done by running `snapper create`. You should always specify a meaningful description with `-d` in order to later be able to identify its purpose.

```
snapper create -d "Before the Apache config cleanup"
```

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

```
snapper -c home create -d ".-file cleanup in ~tux"
```

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

```
snapper -c home create -d "daily HOME backup" -c timeline
```

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

4.4.3 Modifying Snapshot Metadata

Snapper allows to modify the description, the cleanup algorithm and the userdata of a snapshot. All other metadata cannot be changed.

```
snapper modify -c "timeline" 10
```

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

```
snapper -c home modify -d "daily backup" -c "" 120
```

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

4.4.4 Deleting Snapshots

To delete a snapshot, you need to know its number. Get it by running `snapper list`. To delete a snapshot, run `snapper delete NUMBER`. When deleting a pre snapshot, you should always delete its corresponding post snapshot (and vice-versa).

```
snapper delete 65
```

Deletes snapshot 65 for the default (`root`) configuration.

```
snapper -c home delete 89 90
```

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

```
snapper delete 65  
snapper -c myconfig delete 89 90
```

ASTUCE : Old Snapshots Occupy More Disk Space

If you delete snapshots in order to free space on your harddisk (see Section 4.1.1, « snapshots and Disk Space » (p. 99) for details), make sure to delete old snapshots first. The older a snapshot is, the more disk space it occupies.

Snapshots are also automatically deleted by a daily cron-job. Refer to Section 4.4.1.2, « Cleanup-algorithms » (p. 107) for details.

4.5 Disabling Snapper

If you have set up the root partition with `Btrfs` during the installation, Snapper automatically creates hourly snapshots of the system, as well as pre- and post-snapshots for YaST and zypper transactions. Each of these tasks can be disabled as follows:

Disabling Hourly Snapshots

Edit `/etc/snapper/configs/root` and set `TIMELINE_CREATE` to `no`:

```
TIMELINE_CREATE="no"
```

Disabling zypper snapshots

Uninstall the package `snapper-zypp-plugin`

Disabling YaST snapshots

Edit `/etc/sysconfig/yast2` and set `USE_SNAPPER` to `no`:

```
USE_SNAPPER="no"
```

4.6 Creating Snapper Configurations

The way Snapper behaves is defined in a config file that is specific for each partition or `Btrfs` subvolume. These config files reside under `/etc/snapper/configs/`. The default config installed with Snapper for the `/` directory is named `root`. It creates and manages the YaST and zypper snapshots as well as the hourly backup snapshot for `/`.

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

You can use either `snapper` itself or the YaST *Snapper* module to restore files from these snapshots. In YaST you need to select your *Current Configuration*, while you need to specify your config for `snapper` with the global switch `-c` (e.g. `snapper -c myconfig list`).

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

```
snapper -c www-data❶ create-config  
/srv/www❷
```

- ❶ Name of config file.
- ❷ Mount point of the partition or `Btrfs` subvolume to snapshot.

This command will create a new config file `/etc/snapper/config-templates/www-data` with reasonable default values (taken from `/etc/snapper/config-templates/default`).

ASTUCE : Config Defaults

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

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

4.6.1 Adjusting the Config File

To adjust the config file, you need to modify it with an editor. It contains key/value pairs in the form of *key=value*. You may only change the *value*.

SUBVOLUME

Mount point of the partition or subvolume to snapshot. Do not change.

FSTYPE

Filesystem type of the partition. Do not change.

NUMBER_CLEANUP

Defines whether to automatically delete old snapshots when the total snapshot count exceeds a number specified with `NUMBER_LIMIT` *and* an age specified with `NUMBER_MIN_AGE`. Valid values: `yes`, `no`

REMARQUE : Limit and Age

`NUMBER_LIMIT` and `NUMBER_MIN_AGE` are always evaluated both. Snapshots are only deleted when *both* conditions are met. If you always want to keep a certain number of snapshots regardless of their age, set `NUMBER_MIN_AGE` to 0. On the other hand, if you do not want to keep snapshots beyond a certain age, set `NUMBER_LIMIT` to 0.

NUMBER_LIMIT

Defines how many snapshots to keep if `NUMBER_CLEANUP` is set to `yes`.

NUMBER_MIN_AGE

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

`TIMELINE_CREATE`

If set to `yes`, hourly snapshots are created. This is currently the only way to automatically create snapshots, therefore setting it to `yes` is strongly recommended.

Valid values: `yes`, `no`

`TIMELINE_CLEANUP`

Defines whether to automatically delete old snapshots when the snapshot count exceeds a number specified with the `TIMELINE_LIMIT_*` options *and* an age specified with `TIMELINE_MIN_AGE`. Valid values: `yes`, `no`

`TIMELINE_MIN_AGE`

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

`TIMELINE_LIMIT_HOURLY`, `TIMELINE_LIMIT_DAILY`, `TIMELINE_LIMIT_MONTHLY`, `TIMELINE_LIMIT_YEARLY`

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

Example 4.1 Example timeline configuration

```
TIMELINE_CREATE="yes"
TIMELINE_CLEANUP="yes"
TIMELINE_MIN_AGE="1800"
TIMELINE_LIMIT_HOURLY="10"
TIMELINE_LIMIT_DAILY="10"
TIMELINE_LIMIT_MONTHLY="10"
TIMELINE_LIMIT_YEARLY="10"
```

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

Snapshots to be Kept

- Hourly: The last ten snapshots that have been made
- Daily: The first daily snapshot that has been made is kept for the last ten days.
- Monthly: The first snapshot made on the last day of the month is kept for the last ten months.

- Yearly: The first snapshot made on Dec 31st is kept for the last ten years.

4.7 Limitations

Although being ready for production, `Btrfs` as well as Snapper are constantly developed further. The following limitations exist at the moment. It is planned to solve these issues in future releases.

4.7.1 Data Consistency

There is no mechanism to ensure data consistency when creating snapshot. Whenever a file is written (e.g. a database) at the same time the snapshot is created, it will result in a broken or partly written file. Restoring such a file will cause problems. Therefore it is strongly recommended to *always* closely review the list of changed files and their diffs. Only restore files that really need to belonging to the action you want to roll back.

4.7.2 Reverting User Additions

Usually `/home` resides on a separate partition. Such a separate partition is not part of the default configuration for doing YaST rollbacks. Therefore the user's home partition will not be deleted when reverting a user addition using Snapper. It is strongly recommended to use the YaST *User and Group Management* tool to remove users.

4.7.3 No Rollback on `/boot` and Boot Loader Changes

Currently openSUSE cannot boot from `Btrfs` partitions. Therefore a separate partition for `/boot` is created upon the installation when using `Btrfs` for the system partition. Since `/boot` does not support snapshots, the following restrictions apply for YaST/zypper rollbacks:

no rollback for any configuration changes on the bootloader

The only file that can be rolled back is the bootloader configuration file in `/etc`.

The main configuration files reside under `/boot` and cannot be rolled back.

no complete rollback for Kernel installations

The Kernel itself and its `initrd` are installed in the `/boot` partition, whereas Kernel modules or sources are installed in `/var/lib` and `/usr/src`, respectively. Whatsmore, each Kernel installation also changes the bootloader configuration files in `/boot`. So whenever you do a rollback that involves undoing a Kernel installation, you need to manually remove the Kernel and its `initrd` from `/boot` and adjust the bootloader configuration by removing the boot entry for the Kernel.

4.8 Frequently Asked Questions

Why does Snapper never show changes in `/var/log`, `/tmp` and other directories?

For some directories we decided to disable snapshotting, e.g. `/var/log` since reverting logs makes searching for problems difficult. To exclude a path from snapshotting we create a subvolume for that path. The following mount points are excluded from snapshotting on openSUSE:

- `/opt`
- `/srv`
- `/tmp`
- `/var/crash`
- `/var/log`
- `/var/run`
- `/var/spool`
- `/var/tmp`

Can I boot a Snapshot from the Bootloader?

This is currently not possible. The bootloader on openSUSE currently does not support booting from a `Btrfs` partition.

Remote Access with VNC

Virtual Network Computing (VNC) enables you to control a remote computer via a graphical desktop (as opposed to a remote shell access). VNC is platform-independent and lets you access the remote machine from any operating system.

openSUSE supports two different kinds of VNC sessions: One-time sessions that « live » as long as the VNC connection from the client is kept up, and persistent sessions that « live » until they are explicitly terminated.

REMARQUE : Session Types

A machine can offer both kinds of sessions simultaneously on different ports, but an open session cannot be converted from one type to the other.

5.1 One-time VNC Sessions

A one-time session is initiated by the remote client. It starts a graphical login screen on the server. This way you can choose the user which starts the session and, if supported by the login manager, the desktop environment. Once you terminate the client connection to such a VNC session, all applications started within that session will be terminated, too. One-time VNC sessions cannot be shared, but it is possible to have multiple sessions on a single host at the same time.

Procédure 5.1 *Enabling One-time VNC Sessions*

- 1 Start *YaST > Network Services > Remote Administration (VNC)*.

- 2 Check *Allow Remote Administration*.
- 3 If necessary, also check *Open Port in Firewall* (for example, when your network interface is configured to be in the External Zone). If you have more than one network interface, restrict opening the firewall ports to a specific interface via *Firewall Details*.
- 4 Confirm your settings with *Finish*.
- 5 In case not all needed packages are available yet, you need to approve the installation of missing packages.

REMARQUE : Available Configurations

The default configuration on openSUSE serves sessions with a resolution of 1024x768 pixels at a color depth of 16-bit. The sessions are available on ports 5901 for « regular » VNC viewers (equivalent to VNC display 1) and on port 5801 for Web browsers.

Other configurations can be made available on different ports.

VNC display numbers and X display numbers are independent in one-time sessions. A VNC display number is manually assigned to every configuration that the server supports (:1 in the example above). Whenever a VNC session is initiated with one of the configurations, it automatically gets a free X display number.

5.1.1 Initiating a One-time VNC Session

To initiate a one-time VNC session, a VNC viewer must be installed on the client machine. The standard viewer on SUSE Linux products is `vncviewer`, provided by the package `tightvnc`. You may also view a VNC session using your Web browser and a Java applet.

To start your VNC viewer and initiate a session with the server's default configuration, use the command:

```
vncviewer jupiter.example.com:1
```

Instead of the VNC display number you can also specify the port number with two colons:

```
vncviewer jupiter.example.com::5901
```

Alternatively use a Java-capable Web browser to view the VNC session by entering the following URL: `http://jupiter.example.com:5801`

5.1.2 Configuring One-time VNC Sessions

You can skip this section, if you do not need or want to modify the default configuration.

One-time VNC sessions are started via the `xinetd` daemon. A configuration file is located at `/etc/xinetd.d/vnc`. By default it offers six configuration blocks: three for VNC viewers (`vnc1` to `vnc3`), and three serving a Java applet (`vnchttpd1` to `vnchttpd3`). By default only `vnc1` and `vnchttpd1` are active.

To activate a configuration, comment the line `disable = yes` with a `#` character in the first column, or remove that line completely. To deactivate a configuration uncomment or add that line.

The `Xvnc` server can be configured via the `server_args` option—see `Xvnc --help` for a list of options.

When adding custom configurations, make sure they are not using ports that are already in use by other configurations, other services, or existing persistent VNC sessions on the same host.

Activate configuration changes by entering the following command:

```
rcxinetd reload
```

IMPORTANT : Firewall and VNC Ports

When activating Remote Administration as described in Procédure 5.1, « Enabling One-time VNC Sessions » (p. 115), the ports 5801 and 5901 are opened in the firewall. If the network interface serving the VNC sessions is protected by a firewall, you need to manually open the respective ports when activating additional ports for VNC sessions. See Chapitre 13, *Masquerading and Firewalls* (↑Security Guide) for instructions.

5.2 Persistent VNC Sessions

A persistent VNC session is initiated on the server. The session and all applications started in this session run regardless of client connections until the session is terminated.

A persistent session can be accessed from multiple clients simultaneously. This is ideal for demonstration purposes where one client has full access and all other clients have view-only access. Another usecase are trainings where the trainer might need access to the trainee's desktop. However, most of the times you probably do not want to share your VNC session.

In contrast to one-time sessions that start a display manager, a persistent session starts a ready-to operate desktop that runs as the user that started the VNC session.

Access to persistent sessions is protected by two possible types of passwords:

- a regular password that grants full access or
- an optional view-only password that grants a non-interactive (view-only) access.

A session can have multiple client connections of both kinds at once.

Procédure 5.2 *Starting a Persistent VNC Session*

- 1** Open a shell and make sure you are logged in as the user that should own the VNC session.
- 2** If the network interface serving the VNC sessions is protected by a firewall, you need to manually open the port used by your session in the firewall. If starting multiple sessions you may alternatively open a range of ports. See Chapitre 13, *Masquerading and Firewalls* (↑Security Guide) for details on how to configure the firewall.

`vncserver` uses the ports 5901 for display :1, 5902 for display :2, and so on. For persistent sessions, the VNC display and the X display usually have the same number.

- 3** To start a session with a resolution of 1024x769 pixel and with a color depth of 16-bit, enter the following command:

```
vncserver -geometry 1024x768 -depth 16
```


The `vncserver` command picks an unused display number when none is given and prints out its choice. See `man 1 vncserver` for more options.

When running `vncviewer` for the first time, it asks for a password for full access to the session. If needed, you can also provide a password for view-only access to the session.

The password(s) you are providing here are also used for future sessions started by the same user. They can be changed with the `vncpasswd` command.

IMPORTANT : Security Considerations

Make sure to use strong passwords of significant length (eight or more characters). Do not share these passwords.

VNC connections are unencrypted, so people who can sniff the network(s) between the two machines can read the password when it gets transferred at the beginning of a session.

To terminate the session shut down the desktop environment that runs inside the VNC session from the VNC viewer as you would shut it down if it was a regular local X session.

If you prefer to manually terminate a session, open a shell on the VNC server and make sure you are logged in as the user that owns the VNC session you want to terminate. Run the following command to terminate the session that runs on display `:1`:

```
vncserver -kill :1
```

5.2.1 Connecting to a Persistent VNC Session

To connect to a persistent VNC session, a VNC viewer must be installed. The standard viewer on SUSE Linux products is `vncviewer`, provided by the package `tightvnc`. You may also view a VNC session using your Web browser and a Java applet.

To start your VNC viewer and connect to display `:1` of the VNC server, use the command

```
vncviewer jupiter.example.com:1
```

Instead of the VNC display number you can also specify the port number with two colons:

```
vncviewer jupiter.example.com::5901
```

Alternatively use a Java-capable Web browser to view the VNC session by entering the following URL: `http://jupiter.example.com:5801`

5.2.2 Configuring Persistent VNC Sessions

Persistent VNC sessions can be configured by editing `$HOME/.vnc/xstartup`. By default this shell script starts an `xterm` and the `twm` Window Manager. To start either GNOME or KDE instead, replace the line starting `twm` with one of the following:

```
/usr/bin/gnome      # GNOME
/usr/bin/startkde    # KDE
```

REMARQUE : One Configuration for Each User

Persistent VNC sessions are configured in a single per-user configuration. Multiple sessions started by a user will all use the same startup and password files.

Partie III. Système

Applications 32 bits et 64 bits dans un environnement système 64 bits

openSUSE® is available for 64-bit platforms. This does not necessarily mean that all the applications included have already been ported to 64-bit platforms. openSUSE supports the use of 32-bit applications in a 64-bit system environment. This chapter offers a brief overview of how this support is implemented on 64-bit openSUSE platforms. It explains how 32-bit applications are executed (runtime support) and how 32-bit applications should be compiled to enable them to run both in 32-bit and 64-bit system environments. Additionally, find information about the kernel API and an explanation of how 32-bit applications can run under a 64-bit kernel.

openSUSE for the 64-bit platforms amd64 and Intel 64 is designed so that existing 32-bit applications run in the 64-bit environment « out-of-the-box. » This support means that you can continue to use your preferred 32-bit applications without waiting for a corresponding 64-bit port to become available.

6.1 Support d'exécution

IMPORTANT : Conflit entre versions d'application

Si une application est disponible pour les environnements 32 bits et 64 bits, l'installation parallèle des deux versions entraînera certainement des problèmes. Dans un tel cas, choisissez l'une des deux versions et installez uniquement celle-ci.

PAM (Pluggable Authentication Modules) est une exception à cette règle. openSUSE utilise PAM dans le processus d'authentification en tant que

couche intermédiaire entre l'utilisateur et l'application. Sur un système d'exploitation 64 bits qui gère aussi des applications 32 bits, il est nécessaire d'installer les deux versions d'un module PAM.

Pour être correctement exécutée, chaque application nécessite une série de bibliothèques. Malheureusement, les noms des versions 32 bits et 64 bits de ces bibliothèques sont identiques. Elles doivent être différenciées les unes des autres d'une autre façon.

Pour conserver la compatibilité avec la version 32 bits, les bibliothèques sont stockées au même endroit que dans l'environnement 32 bits. La version 32 bits de `libc.so.6` se trouve dans `/lib/libc.so.6` dans les environnements 32 bits et 64 bits.

Toutes les bibliothèques 64 bits et les fichiers objets sont situés dans des répertoires appelés `lib64`. Les fichiers objets 64 bits que vous vous attendriez normalement à trouver dans `/lib` et `/usr/lib` se trouvent maintenant dans `/lib64` et `/usr/lib64`. Cela signifie qu'il y a de la place pour les bibliothèques 32 bits dans `/lib` et `/usr/lib`, le nom de fichier pour les deux versions peut donc rester inchangé.

Subdirectories of 32-bit `/lib` directories which contain data content that does not depend on the word size are not moved. This scheme conforms to LSB (Linux Standards Base) and FHS (File System Hierarchy Standard).

6.2 Développement logiciel

A biarch development tool chain allows generation of 32-bit and 64-bit objects. The default is to compile 64-bit objects. It is possible to generate 32-bit objects by using special flags. For GCC, this special flag is `-m32`.

Tous les fichiers d'en-tête doivent être écrits dans une forme indépendante de l'architecture. Les bibliothèques 32 bits et 64 bits installées doivent avoir une API (Application Programming Interface) qui corresponde avec les fichiers d'en-tête installés. L'environnement openSUSE normal est conçu selon ce principe. Dans le cas de bibliothèques mises à jour manuellement, résolvez ces problèmes vous-même.

6.3 Compilation logicielle sur plateformes biarch

To develop binaries for the other architecture on a biarch architecture, the respective libraries for the second architecture must additionally be installed. These packages are called `rpmname-32bit`. You also need the respective headers and libraries from the `rpmname-devel` packages and the development libraries for the second architecture from `rpmname-devel-32bit`.

La plupart des programmes open source utilisent une configuration de programme basée sur `autoconf`. Pour utiliser `autoconf` pour configurer un programme pour la seconde architecture, écrasez les paramètres normaux du compilateur et du linker de `autoconf` en exécutant le script `configure` avec les variables d'environnement additionnelles.

L'exemple suivant fait référence à un système `x86_64` avec `x86` comme seconde architecture.

1 Utiliser le compilateur 32 bits :

```
CC="gcc -m32"
```

2 Instruct the linker to process 32-bit objects (always use `gcc` as the linker front-end):

```
LD="gcc -m32"
```

3 Paramétrer l'assembleur pour générer des objets 32 bits :

```
AS="gcc -c -m32"
```

4 Spécifier les options du linker, comme par exemple l'emplacement des bibliothèques 32 bits :

```
LDFLAGS="-L/usr/lib"
```

5 Specify the location for the 32-bit object code libraries:

```
--libdir=/usr/lib
```

6 Spécifier l'emplacement pour les bibliothèques X 32 bits :

```
--x-libraries=/usr/lib
```

Ces variables ne sont pas toutes nécessaires pour chaque programme. Adaptez-les au programme concerné.

```
CC="gcc -m32"  
LDFLAGS="-L/usr/lib;"  
./configure --prefix=/usr --libdir=/usr/lib --x-libraries=/usr/lib  
make  
make install
```

6.4 Spécifications du noyau

Les noyaux 64 bits pour x86_64 offrent une ABI noyau (Application Binary Interface) 64 bits et 32 bits. Cette dernière est identique à l'ABI correspondante au noyau 32 bits. Cela signifie que l'application 32 bits peut communiquer avec le noyau 64 bits de la même façon qu'avec le noyau 32 bits.

The 32-bit emulation of system calls for a 64-bit kernel does not support all the APIs used by system programs. This depends on the platform. For this reason, a small number of applications, like `lspci`, must be compiled.

Un noyau 64 bits ne peut charger que des modules noyau 64 bits qui ont été spécialement compilés pour ce noyau. Il n'est pas possible d'utiliser des modules noyau 32 bits.

ASTUCE : Kernel-loadable Modules

Some applications require separate kernel-loadable modules. If you intend to use such a 32-bit application in a 64-bit system environment, contact the provider of this application and SUSE to make sure that the 64-bit version of the kernel-loadable module and the 32-bit compiled version of the kernel API are available for this module.

Booting a Linux System

Booting a Linux system involves different components and tasks. The hardware itself is initialized by the BIOS or the UEFI, which starts the Kernel by means of a boot loader. After this point, the boot process is completely controlled by the operating system and handled by `systemd`. `systemd` provides a set of « targets » that boot setups for everyday usage, maintenance or emergencies.

7.1 The Linux Boot Process

The Linux boot process consists of several stages, each represented by a different component. The following list briefly summarizes the boot process and features all the major components involved:

1. **BIOS/UEFI** After turning on the computer, the BIOS or the UEFI initializes the screen, and keyboard and tests the main memory. Up to this stage, the machine does not access any mass storage media. Subsequently, the information about the current date, time, and the most important peripherals are loaded from the CMOS values. When the first hard disk and its geometry are recognized, the system control passes from the BIOS to the boot loader. If the BIOS supports network booting, it is also possible to configure a boot server that provides the boot loader. On x86 systems, PXE boot is needed. Other architectures commonly use the BOOTP protocol to get the boot loader.
2. **Boot Loader** The first physical 512-byte data sector of the first hard disk is loaded into the main memory and the *boot loader* that resides at the beginning of

this sector takes over. The commands executed by the boot loader determine the remaining part of the boot process. Therefore, the first 512 bytes on the first hard disk are referred to as the *Master Boot Record* (MBR). The boot loader then passes control to the actual operating system, in this case, the Linux Kernel. More information about GRUB, the Linux boot loader, can be found in Chapitre 9, *The Boot Loader GRUB* (p. 153). For a network boot, the BIOS acts as the boot loader. It gets the boot image from the boot server and starts the system. This is completely independent of local hard disks.

3. **Kernel and `initramfs`** To pass system control, the boot loader loads both the Kernel and an initial RAM-based file system (`initramfs`) into memory. The contents of the `initramfs` can be used by the Kernel directly. `initramfs` contains a small executable called `init` that handles the mounting of the real root file system. If special hardware drivers are needed before the mass storage can be accessed, they must be in `initramfs`. For more information about `initramfs`, refer to Section 7.1.1, « `initramfs` » (p. 129). If the system does not have a local hard disk, the `initramfs` must provide the root file system for the Kernel. This can be done with the help of a network block device like iSCSI or SAN, but it is also possible to use NFS as the root device.

REMARQUE : The `init` Process Naming

Two different programs are commonly named « `init` »:

- a. the `initramfs` process mounting the root filesystem
- b. the operating system process setting up the system

In this chapter we will therefore refer to them as « `init on initramfs` » and « `systemd` », respectively.

4. **`init on initramfs`** This program performs all actions needed to mount the proper root file system. It provides Kernel functionality for the needed file system and device drivers for mass storage controllers with `udev`. After the root file system has been found, it is checked for errors and mounted. If this is successful, the `initramfs` is cleaned and the `systemd` daemon on the root file system is executed. For more information about `init on initramfs`, refer to Section 7.1.2, « `init on initramfs` » (p. 130). Find more information about `udev` in Chapitre 12, *Dynamic Kernel Device Management with `udev`* (p. 203).

5. **systemd** By starting services and mounting file systems `systemd` handles the actual booting of the system. `systemd` is described in Chapitre 8, *The systemd daemon* (p. 133).

7.1.1 `initramfs`

`initramfs` is a small `cpio` archive that the Kernel can load into a RAM disk. It provides a minimal Linux environment that enables the execution of programs before the actual root file system is mounted. This minimal Linux environment is loaded into memory by BIOS or UEFI routines and does not have specific hardware requirements other than sufficient memory. The `initramfs` archive must always provide an executable named `init` that executes the `systemd` daemon on the root file system for the boot process to proceed.

Before the root file system can be mounted and the operating system can be started, the Kernel needs the corresponding drivers to access the device on which the root file system is located. These drivers may include special drivers for certain kinds of hard drives or even network drivers to access a network file system. The needed modules for the root file system may be loaded by `init` on `initramfs`. After the modules are loaded, `udev` provides the `initramfs` with the needed devices. Later in the boot process, after changing the root file system, it is necessary to regenerate the devices. This is done by the `systemd` unit `udev.service` with the command `udevtrigger`.

If you need to change hardware (for example hard disks) in an installed system and this hardware requires different drivers to be present in the Kernel at boot time, you must update `initramfs`. This is done by calling `mkinitrd`. Calling `mkinitrd` without any argument creates an `initramfs`. Calling `mkinitrd -R` creates an `init` executable. In openSUSE®, the modules to load are specified by the variable `INITRD_MODULES` in `/etc/sysconfig/kernel`. After installation, this variable is automatically set to the correct value. The modules are loaded in exactly the order in which they appear in `INITRD_MODULES`.

IMPORTANT : Updating `initramfs` or `init`

The boot loader loads `initramfs` or `init` in the same way as the Kernel. It is not necessary to re-install GRUB after updating `initramfs` or `init`, because GRUB searches the directory for the right file when booting.

7.1.2 init on initramfs

The main purpose of `init` on `initramfs` is to prepare the mounting of and access to the real root file system. Depending on your system configuration, `init` on `initramfs` is responsible for the following tasks.

Loading Kernel Modules

Depending on your hardware configuration, special drivers may be needed to access the hardware components of your computer (the most important component being your hard drive). To access the final root file system, the Kernel needs to load the proper file system drivers.

Providing Block Special Files

For each loaded module, the Kernel generates device events. `udev` handles these events and generates the required special block files on a RAM file system in `/dev`. Without those special files, the file system and other devices would not be accessible.

Managing RAID and LVM Setups

If you configured your system to hold the root file system under RAID or LVM, `init` on `initramfs` sets up LVM or RAID to enable access to the root file system later. Find information about RAID and LVM in Chapitre 3, *Configuration avancée des disques* (p. 75).

Managing Network Configuration

If you configured your system to use a network-mounted root file system (mounted via NFS), `init` on `initramfs` must make sure that the proper network drivers are loaded and that they are set up to allow access to the root file system.

If the file system resides on a network block device like iSCSI or SAN, the connection to the storage server is also set up by `init` on `initramfs`.

When `init` on `initramfs` is called during the initial boot as part of the installation process, its tasks differ from those mentioned above:

Finding the Installation Medium

When starting the installation process, your machine loads an installation Kernel and a special `init` containing the YaST installer. The YaST installer is running in a RAM file system and needs to have information about the location of the installation medium in order to access it for installing the operating system.

Initiating Hardware Recognition and Loading Appropriate Kernel Modules

As mentioned in Section 7.1.1, « `initramfs` » (p. 129), the boot process starts with a minimum set of drivers that can be used with most hardware configurations. `init` starts an initial hardware scanning process that determines the set of drivers suitable for your hardware configuration. The names of the modules needed for the boot process are written to `INITRD_MODULES` in `/etc/sysconfig/kernel`. These names are used to generate a custom `initramfs` that is needed to boot the system. If the modules are not needed for boot but for coldplug, the modules are written to `/etc/sysconfig/hardware/hwconfig-*`. All devices that are described with configuration files in this directory are initialized in the boot process.

Loading the Installation System

As soon as the hardware is properly recognized, the appropriate drivers are loaded. The `udev` program creates the special device files and `init` starts the installation system with the YaST installer.

Starting YaST

Finally, `init` starts YaST, which starts package installation and system configuration.

The `systemd` daemon

The program `systemd` is the process with process ID 1. It is responsible for initializing the system in the required way. `systemd` is started directly by the Kernel and resists signal 9, which normally kills processes. All other programs are either started directly by `systemd` or by one of its child processes.

REMARQUE : Using System V init rather than `systemd`

Being fully compatible, both `systemd` and System V init (SysV init) can act as a drop-in replacement for each other. If you prefer to use the SysV init rather than `systemd`, press F5 on the boot screen and choose *System V*. To make this a permanent choice, install the package `sysvinit-init` which will replace the `systemd-init` package.

Starting with openSUSE 12 `systemd` is a replacement for the popular SysV init daemon. `systemd` is fully compatible with SysV init (by supporting init scripts). One of the main advantages of `systemd` is the fact that it considerably speeds up boot time by aggressively paralleling service starts. Furthermore, `systemd` only starts a service when it is really needed. For example, the printing daemon `cupsd` is not started during boot time, but rather when a user prints a document for the first time after having booted the system. `systemd` also supports Kernel Control Groups (cgroups), snapshotting and restoring the system state and more. See <http://www.freedesktop.org/wiki/Software/systemd/> for details.

8.1 Basic Usage

The SysV init system utilized several different commands to handle services— the `init` scripts, `insserv`, `telinit` and others. `systemd` makes it easier to manage services, since there is only one command to memorize for the majority of service handling tasks: `systemctl`. It uses the command plus subcommand notation like `git` or `zypper`:

```
systemctl [general OPTIONS] subcommand [subcommand OPTIONS]
```

See `man 1 systemctl` for a complete manual.

ASTUCE : Terminal Output and Bash Completion

If the output goes to a terminal (and not to a pipe or a file, for example) `systemd` commands send long output to a pager by default. Use the `--no-pager` option to turn off paging mode.

`systemd` also supports bash-completion, allowing you to enter the first letters of a subcommand and then hit `→` to automatically complete it. This feature is only available in the `bash` Shell and requires the installation of the package `bash-completion`.

8.1.1 Managing Services in a Running System

Subcommands for managing services are the same as for managing a service with SysV init (`start`, `stop`, ...). The general syntax for service management commands is as follows:

`systemd`

```
systemctl reload|restart|start|status|stop|... <my_service(s)>.service
```

SysV init

```
rc<my_service(s)> reload|restart|start|status|stop|...
```

`systemd` allows to manage several services in one go. Instead of executing init scripts one after the other as with SysV init, execute a command like the following:

```
systemctl start <my_1st_service>.service <my_2nd_service>.service
```


The following table shows a list of the most important service management commands for `systemd` and SysV `init`:

Tableau 8.1 *Service Management Commands*

Task	systemd Subcom- mand	SysV init Subcom- mand
Starting	<code>start</code>	<code>start</code>
Stopping	<code>stop</code>	<code>stop</code>
Restarting <i>Shuts down services and starts them afterwards. If a service is not yet running it will be started.</i>	<code>restart</code>	<code>restart</code>
Restarting conditionally <i>Restarts services if they are currently running. Does nothing for services that are not running.</i>	<code>try-restart</code>	<code>try-restart</code>
Reloading <i>Tells services to reload their configuration files without interrupting operation. Use case: Tell Apache to reload a modified <code>httpd.conf</code> configuration file. Note that not all services support reloading.</i>	<code>reload</code>	<code>reload</code>
Reloading or restarting <i>Reloads services if reloading is supported, otherwise restarts them. Services is not yet running it will be started.</i>	<code>reload-or-restart</code>	n/a

Task	systemd Subcommand	SysV init Subcommand
Reloading or restarting conditionally <i>Reloads services if reloading is supported, otherwise restarts them if currently running. Does nothing for services that are not running.</i>	<code>reload-or-try-restart</code>	n/a
Getting detailed status information <i>Lists information about the status of services. The <code>systemd</code> command shows details such as description, executable, status, cgroup, and messages last issued by a service (see Section 8.1.2, « Debugging Services » (p. 136)). The level of details displayed with the SysV init differs from service to service.</i>	<code>status</code>	<code>status</code>
Getting short status information <i>Shows whether services are active or not.</i>	<code>is-active</code>	<code>status</code>

8.1.2 Debugging Services

By default, `systemd` is not overly verbose. If a service was started successfully, no output will be produced. In case of a failure, a short error message will be displayed. However, `systemctl status` provides means to debug start-up as well as operation of a service.

`systemd` comes with its own logging mechanism (« The Journal ») that logs system messages. This allows to display the service messages together with status messages. The `status` command works similar to `tail` and can also display the log messages in different formats, making it a powerful debugging tool.

Show Service Start-Up Failure

Whenever a service fails to start, use `systemctl status <my_service>.service` to get a detailed error message:

```
www.example.com: ~ # systemctl start apache2.service
Job failed. See system journal and 'systemctl status' for details.
www.example.com: ~ # systemctl status apache2.service
   Loaded: loaded (/lib/systemd/system/apache2.service; disabled)
   Active: failed (Result: exit-code) since Mon, 04 Jun 2012 16:52:26
+0200; 29s ago
   Process: 3088 ExecStart=/usr/sbin/start_apache2 -D SYSTEMD -k start
(code=exited, status=1/FAILURE)
   CGroup: name=systemd:/system/apache2.service

Jun 04 16:52:26 g144 start_apache2[3088]: httpd2-prefork: Syntax error
on line
205 of /etc/apache2/httpd.conf: Syntax error on li...alHost>
```

Show Last *n* Service Messages

The default behavior of the `status` subcommand is displaying the last ten messages a service issued. To change the number of messages to show, use the `--lines=n` parameter,

```
systemctl status ntp.service
systemctl --lines=20 status ntp.service
```

Show Service Messages in Append Mode

To display a « live stream » of service messages, use the `--follow` option, which works just like `tail -f`:

```
systemctl --follow status ntp.service
```

Messages Output Format

The `--output=mode` parameter allows to change the output format of service messages. The most important modes available are:

`short`

The default format. Shows the log messages with a human readable timestamp.

`verbose`

Full output with all fields.

`cat`

Terse output without time stamps.

8.1.3 Permanently Enabling/Disabling Services

The service management commands mentioned in the previous section let you manipulate services for the current session. `systemd` also lets you permanently enable or disable services, so they are automatically started when requested or are always unavailable. You can either do this by using YaST, or on the command line.

8.1.3.1 Enabling/Disabling Services on the Command Line

The following table shows a list of enabling and disabling commands for `systemd` and SysV init:

IMPORTANT : Service Start

When enabling a service on the command line, it is not started automatically. It is scheduled to be started with the next system start-up or runlevel/target change. In order to immediately start a service after having enabled it, explicitly run `systemctl start <my_service>.service` or `rc<my_service> start`.

Tableau 8.2 *Commands for Enabling and Disabling Services*

Task	systemd subcommand	SysV init subcommand
Enabling	<code>systemctl enable <my_service(s)>.service</code>	<code>insserv <my_service(s)></code>
Disabling	<code>systemctl disable <my_service(s)>.service</code>	<code>insserv -r <my_service(s)></code>
Checking <i>Shows whether a service is enabled or not.</i>	<code>systemctl is-enabled <my_service>.service</code>	n/a

Task	systemd subcommand	SysV init subcommand
Re-Enabling <i>Similar to restarting a service, this commands first disables and then enables a service. Useful to re-enable a service with its defaults.</i>	<code>systemctl reenab <my_service>.service</code>	n/a
Masking <i>When « disabling » a service, it can still be started manually. To completely disable a service, you need to mask it. Use with care.</i>	<code>systemctl mask <my_service>.service</code>	n/a
Unmasking <i>A service that has been masked can only be used again after it has been unmasked.</i>	<code>systemctl unmask <my_service>.service</code>	n/a

8.1.3.2 Enabling/Disabling Services with YaST

Start the YaST module with *YaST > System > System Services (Runlevel)*. In the default view, the *Simple Mode*, it displays an overview listing all the available services and the current status of each service (see Figure 8.1, « System Services (Runlevel) » (p. 140)). The left column shows the name of the service, the center column indicates its current status and the right column gives a short description. For the selected service, a more detailed description is provided in the lower part of the window. To enable a service, select it in the table then select *Enable*. The same steps apply to disable a service.

The *Expert Mode* provides detailed control over the runlevels (refer to Section 8.2.1, « Targets vs. Runlevels » (p. 141) for more information on runlevels). The runlevel into which the system boots by default is displayed at the top. Normally it defaults to runlevel 5 (full multiuser mode with network and X). A suitable alternative might be runlevel 3 (full multiuser mode with network). Runlevel 4 is undefined to allow creation of a custom runlevel.

To enable or disable individual services use the table in this window. It lists the services available, shows whether they are currently enabled on your system and, if so, for which runlevels. To define the runlevels in which a service should be started, select the respective row with the mouse and activate the checkboxes below that represent the runlevels. A brief description of the currently selected service or daemon is provided below the table overview.

AVERTISSEMENT : Faulty Runlevel Settings May Damage Your System

Faulty runlevel settings may make your system unusable. Before applying your changes, make absolutely sure that you know their consequences.

Figure 8.1 *System Services (Runlevel)*



With *Start/Stop/Refresh*, decide whether a service should be activated. *Refresh status* checks the current status. *Set/Reset* lets you enable or disable a service in the same manner as with the *Simple Mode* interface. Selecting *OK* saves the changed settings to disk.

8.2 System Start and Target Management

The entire process of starting the system and shutting it down is maintained by `systemd`. From this point of view, the Kernel can be considered a background process to maintain all other processes and adjust CPU time and hardware access according to requests from other programs.

8.2.1 Targets vs. Runlevels

With SysV `init` the system was booted into a so called « Runlevel ». A runlevel defines how the system is started and what services are available in the running system. Runlevels are numbered, the most commonly known ones are 0 (shutting down the system), 3 (multiuser with network) and 5 (multiuser with network and displaymanager).

`systemd` introduces a new concept by using so-called « target units ». However, it remains fully compatible to the runlevel concept. Target units are named rather than numbered and serve specific purposes. For example, the targets `local-fs.target` and `swap.target` mount local filesystems and swap spaces.

The target `graphical.target` provides a multiuser system with network and displaymanager capabilities and is equivalent to runlevel 5. Complex targets, such as `graphical.target` act as « meta » targets by combining a subset of other targets. Since `systemd` makes it easy to create custom targets by combining existing targets, it offers great flexibility.

The following list shows a list of the most important `systemd` target units. For a full list refer to `man 7 systemd.special`.

Selected `systemd` Target Units

`default.target`

The target that is booted by default. Not a « real » target, but rather a symlink to another target like `graphic.target`. Can be permanently changed via YaST (see Section 8.1.3.2, « Enabling/Disabling Services with YaST » (p. 139)). To change it for a session, use the Kernel command line option `systemd.unit=<my_target>.target` at the boot prompt.

`emergency.target`

Starts an emergency shell on the console. Only use it at the boot prompt as `systemd.unit=emergency.target`.

`graphical.target`

Starts a system with network, multi-user support and a displaymanager.

`halt.target`

Shuts down the system.

`mail-transfer-agent.target`

Starts all services necessary for sending and receiving mails.

`multi-user.target`

Starts a multi-user system with network.

`reboot.target`

Reboots the system.

`rescue.target`

Starts a single user system without network.

In order to remain compatible to the SysV init runlevel system, `systemd` provides special targets named `runlevelX.target` mapping the corresponding runlevels numbered X.

Tableau 8.3 SysV Runlevels and `systemd` Target Units

SysV runlevel	<code>systemd</code> target	Purpose
0	<code>runlevel0.target</code> , <code>halt.target</code> , <code>poweroff.target</code>	System shutdown

SysV runlevel	systemd target	Purpose
1, S	runlevel1.target, rescue.target,	Single user mode
2	runlevel2.target, multi-user.target,	Local multiuser without remote network
3	runlevel3.target, multi-user.target,	Full multiuser with network
4	runlevel4.target	Unused/User Defined
5	runlevel5.target, graphical.target,	Full multiuser with network and displaymanager
6	runlevel6.target, reboot.target,	System reboot

IMPORTANT : systemd ignores /etc/inittab

The runlevels in a SysV init system are configured in `/etc/inittab`. `systemd` does *not* use this configuration. Please refer to Section 8.2.2, « Custom Targets » (p. 145) for instructions on how to create your own bootable target.

8.2.1.1 Commands to Change Targets

Use the following commands to operate with target units:

Task	systemd command	SysV init command
Change the current target/ runlevel	<code>systemctl isolate <my_target>.target</code>	<code>telinit X</code>

Task	systemd command	SysV init command
Change to the default target/ runlevel	<code>systemctl default</code>	n/a
Get the current target/ runlevel	<code>systemctl list-units --type=target</code> With <code>systemd</code> there is usually more than one active target. The command lists all targets currently active.	<code>who -r</code> or <code>runlevel</code>
Change the default runlevel persistently	Use the YaST runlevel editor or run the following command: <pre>ln -sf /lib/systemd/system/<my_target>.target /etc/systemd/system/default.target</pre>	Use the YaST runlevel editor or change the line <code>id:X:initdefault:</code> <code>in /etc/inittab</code>
Change the default runlevel for the current boot process	Enter the following option at the boot prompt <code>systemd.unit=<my_target.target></code>	Enter the desired runlevel number at the boot prompt.
Show a target's/ runlevel's dependencies	<pre>systemctl show -p "Requires" <my_target.target></pre> <pre>systemctl show -p "Wants" <my_target.target></pre> <p>« Requires » lists the hard dependencies (the ones that must be resolved), whereas « Wants » lists the soft</p>	n/a

Task	systemd command	SysV init command
	dependencies (the ones that get resolved if possible).	

8.2.2 Custom Targets

On SysV init SUSE systems, runlevel 4 is unused to allow administrators to create their own runlevel configuration. `systemd` allows to create any number of custom targets. It's suggested to start by adapting an existing target such as `graphical.target`.

AVERTISSEMENT : Avoiding Overwritten Customizations

Always do `systemd` customization in `/etc/systemd`, *never* in `/lib/systemd`. Otherwise your changes will be overwritten by the next update of `systemd`.

Procédure 8.1 *Create a Custom Target*

- 1 Copy the configuration file `/lib/systemd/system/graphical.target` to `/etc/systemd/system/<my_target>.target` and adjust it according to your needs.
- 2 The configuration file copied in the previous step already covers the required (« hard ») dependencies for the target. To also cover the wanted (« soft ») dependencies, create a directory `/etc/systemd/system/<my_target>.target.wants`.
- 3 For each wanted service create a symbolic link from `/lib/systemd/system` into `/etc/systemd/system/<my_target>.target.wants`.
- 4 Once you have finished setting up the target, reload the `systemd` configuration to make the new target available:

```
systemctl daemon-reload
```

8.2.3 Debugging System Start-Up

systemd offers means to analyze the system start-up process. You can conveniently review the list of all services and their status (rather than having to parse /varlog/). systemd also allows to scan the start-up procedure to find out how much time each service start-up consumes.

8.2.3.1 Review Start-Up of Services

To review the complete list of services that have been started since booting the system, just enter the command `systemctl`. It lists all active services like shown below (shortened). To get more information on a specific service, use `systemctl status <my_service>.service`.

Example 8.1 *List Active Services*

```
jupiter.example.com: ~ # systemctl
UNIT                                LOAD    ACTIVE SUB    JOB DESCRIPTION
[...]
systemd-random-seed-load.path      loaded active waiting    Random Seed
acpid.service                      loaded active running    ACPI Event
  Daemon
apache2.service                   loaded failed failed      apache
avahi-daemon.service              loaded active running    Avahi mDNS/
DNS-SD Stack
bluez-coldplug.service            loaded active exited     LSB: handles
  udev coldplug of bluetooth dongles
console-kit...-system-start.service loaded active exited     Console
  System Startup Logging
cron.service                      loaded active running    Command
  Scheduler
cups.service                      loaded active running    CUPS
  Printing Service
[...]
LOAD    = Reflects whether the unit definition was properly loaded.
ACTIVE  = The high-level unit activation state, i.e. generalization of SUB.
SUB     = The low-level unit activation state, values depend on unit type.
JOB     = Pending job for the unit.
```

107 units listed. Pass `--all` to see inactive units, too.

To restrict the output to services that failed to start, use the `--failed` option:

Example 8.2 *List Failed Services*

```
jupiter.example.com: ~ # systemctl --failed
```

UNIT	LOAD	ACTIVE	SUB	JOB	DESCRIPTION
apache2.service	loaded	failed	failed		apache
NetworkManager.service	loaded	failed	failed		Network Manager
plymouth-start.service	loaded	failed	failed		Show Plymouth Boot Screen

[...]

8.2.3.2 Debug Start-Up Time

To debug system start-up time, `systemd` offers the `systemd-analyze` command. It shows the total start-up time, a list of services ordered by start-up time and can also generate an SVG graphic showing the time services took to start in relation to the other services.

Listing the System's Start-Up Time

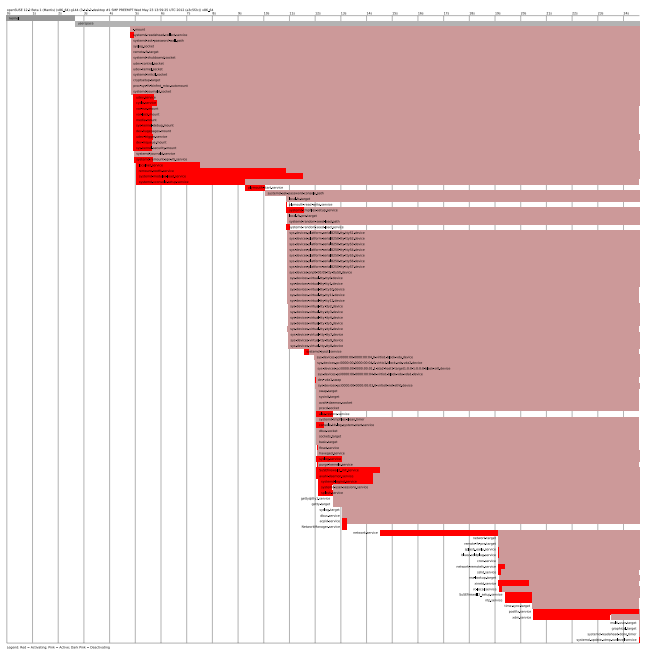
```
jupiter.example.com: ~ # systemd-analyze
Startup finished in 2666ms (kernel) + 21961ms (userspace) = 24628ms
```

Listing the Services Start-Up Time

```
jupiter.example.com: ~ # systemd-analyze blame
6472ms systemd-modules-load.service
5833ms remount-rootfs.service
4597ms network.service
4254ms systemd-vconsole-setup.service
4096ms postfix.service
2998ms xdm.service
2483ms localnet.service
2470ms SuSEfirewall2_init.service
2189ms avahi-daemon.service
2120ms systemd-logind.service
1210ms xinetd.service
1080ms ntp.service
[...]
75ms fbset.service
72ms purge-kernels.service
47ms dev-vda1.swap
38ms bluez-coldplug.service
35ms splash_early.service
```

Services Start-Up Time Graphics

```
jupiter.example.com: ~ # systemd-analyze plot > jupiter.example.com-
startup.svg
```



8.2.3.3 Review the Complete Start-Up Process

Above mentioned commands let you review the services that started and the time it took to start them. If you need to know more details, you can tell `systemd` to verbosely log the complete start-up procedure by entering the following parameters at the boot prompt:

```
systemd.log_level=debug systemd.log_target=kmsg
```

Now `systemd` writes its log messages into the kernel ring buffer. View that buffer with `dmesg`:

```
dmesg | less
```

8.3 Advanced Usage

The following sections cover advanced topics for system administrators. For even more advanced `systemd` documentation, please refer Lennart Pöttering's

series about `systemd` for administrators at <http://0pointer.de/blog/projects>.

8.3.1 System Log

Section 8.1.2, « Debugging Services » (p. 136) explains how to view log messages for given service. However, displaying log messages is not restricted to service logs. You can also access and query the complete log written by `systemd`—the so-called « Journal ». Use the command `systemd-journalctl` to display the complete log starting with the oldest entries. Refer to `man 1 systemd-journalctl` for options such as applying filters or changing the output format.

8.3.2 Snapshots

You can save the current state of `systemd` to a named snapshot and later revert to it with the `isolate` subcommand. This is useful when testing services or custom targets, because it allows you to return to a defined state at any time. A snapshot is only available in the current session and will automatically be deleted on reboot. A snapshot name must end in `.snapshot`.

Create a Snapshot

```
systemctl snapshot <my_snapshot>.snapshot
```

Delete a Snapshot

```
systemctl delete <my_snapshot>.snapshot
```

View a Snapshot

```
systemctl show <my_snapshot>.snapshot
```

Activate a Snapshot

```
systemctl isolate <my_snapshot>.snapshot
```

8.3.3 Kernel Control Groups (cgroups)

On a traditional SysV init system it is not always possible to clearly assign a process to the service that spawned it. Some services such as Apache, spawn a lot of 3rd party

processes (e.g. CGI or Java processes), which themselves spawn more processes. This makes a clear assignment difficult or even impossible. Additionally, a service may not terminate correctly, leaving some of its children alive.

`systemd` solves this problem by placing each service into its own cgroup. cgroups are a Kernel feature that allows aggregating processes and all their children into hierarchical organized groups. `systemd` names each cgroup after its service. Since a non-privileged process is not allowed to « leave » its cgroup, this provides an effective way to label all processes spawned by a service with the name of the service.

To list all processes belonging to a service, use the command `systemd-cgls`. The result will look like the following (shortened) example:

Example 8.3 *List all Processes Belonging to a Service*

```
~ # systemd-cgls --no-pager
├─ user
│   └─ root
│       └─ 1
│           ├── 2279 sshd: root@pts/0
│           ├── 2282 -bash
│           └─ 2541 systemd-cgls --no-pager
└─ system
    ├── 1 /sbin/init splash showopts
    ├── apache2.service
    │   ├── 2535 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D SYSTEMD -
    │   k start
    │   ├── 2536 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D SYSTEMD -
    │   k start
    │   ├── 2537 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D SYSTEMD -
    │   k start
    │   ├── 2538 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D SYSTEMD -
    │   k start
    │   ├── 2539 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D SYSTEMD -
    │   k start
    │   └─ 2540 /usr/sbin/httpd2-prefork -f /etc/apache2/httpd.conf -D SYSTEMD -
    │   k start
    ├── xdm.service
    │   ├── 2250 /usr/bin/xdm
    │   ├── 2253 /usr/bin/X -nolisten tcp -br vt7 -auth /var/lib/xdm/authdir/
    │   authfiles/A:0-ii8Goo
    │   ├── 2263 -:0
    │   └─ 2276 /usr/bin/xconsole -notify -nostdin -verbose -exitOnFail
    ├── ntp.service
    │   └─ 2202 /usr/sbin/ntpd -p /var/run/ntp/ntpd.pid -g -u ntp:ntp -c /etc/
    ntp.conf
    ├── sshd.service
    │   └─ 1743 /usr/sbin/sshd -D
```


See Chapitre 10, *Kernel Control Groups* (↑System Analysis and Tuning Guide) for more information about cgroups.

8.3.4 Killing Services (Sending Signals)

As explained in Section 8.3.3, « Kernel Control Groups (cgroups) » (p. 149), it is not always possible to assign a process to its parent service process in a SysV init system. This makes it difficult to terminate a service and all of its children. Child processes that have not been killed will remain as zombie processes.

`systemd`'s concept of to confine each service into a cgroup makes it possible to clearly identify all child processes of a service and therefore allows to send a signal to each of these processes. Use `systemctl kill` to send signals to services. For a list of available signals refer to `man 7 signals`.

Sending SIGTERM to a Service

`SIGTERM` is the default signal that is sent.

```
systemctl kill <my_service>.service
```

Sending *SIGNAL* to a Service

Use the `-s` option to specify the signal that should be sent.

```
systemctl kill -s SIGNAL <my_service>.service
```

Selecting Processes

By default the `kill` command send the signal to all processes of the specified cgroup. You can restrict it to the `control` or the `main` process. The latter is for example useful to force a service to reload its configuration by sending `SIGHUP`:

```
systemctl kill -s SIGHUP --kill-who=main <my_service>.service
```

8.4 More information

For more information on `systemd` refer to the following online resources:

Homepage

<http://www.freedesktop.org/wiki/Software/systemd>

`systemd` for Administrators

Lennart Pöttering, one of the `systemd` authors, has written a series of blog entries (13 at the time of writing this chapter). Find them at <http://0pointer.de/blog/projects>.

Control Centre: The systemd Linux init system

<http://www.h-online.com/open/features/Control-Centre-The-systemd-Linux-init-system-1565543.html>

Booting up: Tools and tips for systemd, a Linux init tool

<http://www.h-online.com/open/features/Booting-up-Tools-and-tips-for-systemd-1570630.html>

The Boot Loader GRUB

This chapter describes how to configure GRUB (Grand Unified Bootloader), the boot loader used in openSUSE®. A special YaST module is available for configuring all settings. If you are not familiar with the subject of booting in Linux, read the following sections to acquire some background information. This chapter also describes some of the problems frequently encountered when booting with GRUB and their solutions.

This chapter focuses on boot management and the configuration of the boot loader GRUB. The boot procedure as a whole is outlined in Chapitre 7, *Booting a Linux System* (p. 127). A boot loader represents the interface between the machine (BIOS) and the operating system (openSUSE). The configuration of the boot loader directly impacts the start of the operating system.

The following terms appear frequently in this chapter and might need some explanation:

MBR (Master Boot Record)

The structure of the MBR is defined by an operating system-independent convention. The first 446 bytes are reserved for the program code. They typically hold part of a boot loader program or an operating system selector. The next 64 bytes provide space for a partition table of up to four entries. The partition table contains information about the partitioning of the hard disk and the file system types. The operating system needs this table for handling the hard disk. With conventional generic code in the MBR, exactly one partition must be marked *active*. The last two bytes of the MBR must contain a static « magic

number » (AA55). An MBR containing a different value is regarded as invalid by some BIOSes, so is not considered for booting.

Boot Sectors

Boot sectors are the first sectors of hard disk partitions with the exception of the extended partition, which merely serves as a « container » for other partitions. These boot sectors have 512 bytes of space for code used to boot an operating system installed in the respective partition. This applies to boot sectors of formatted DOS, Windows, and OS/2 partitions, which also contain some basic important data of the file system. In contrast, the boot sectors of Linux partitions are initially empty after setting up a file system other than XFS. Therefore, a Linux partition is not bootable by itself, even if it contains a kernel and a valid root file system. A boot sector with valid code for booting the system has the same magic number as the MBR in its last two bytes (AA55).

9.1 Booting with GRUB

GRUB comprises two stages. Stage 1 consists of 512 bytes and its only task is to load the second stage of the boot loader. Subsequently, stage 2 is loaded. This stage contains the main part of the boot loader.

In some configurations, an intermediate stage 1.5 can be used, which locates and loads stage 2 from an appropriate file system. If possible, this method is chosen by default on installation or when initially setting up GRUB with YaST.

Stage 2 is able to access many file systems. Currently, ext2, ext3, ReiserFS, Minix, and the DOS FAT file system used by Windows are supported. To a certain extent, XFS, and UFS and FFS used by BSD systems are also supported. Since version 0.95 GRUB is also able to boot from a CD or DVD containing an ISO 9660 standard file system pursuant to the « El Torito » specification. Even before the system is booted, GRUB can access file systems of supported BIOS disk devices (floppy disks or hard disks, CD drives and DVD drives detected by the BIOS). Therefore, changes to the GRUB configuration file (`menu.lst`) do not require a new installation of the boot manager. When the system is booted, GRUB reloads the menu file with the valid paths and partition data of the kernel or the initial RAM disk (`initrd`) and locates these files.

The actual configuration of GRUB is based on four files that are described below:

`/boot/grub/menu.lst`

This file contains all information about partitions or operating systems that can be booted with GRUB. Without this information, the GRUB command line prompts the user for how to proceed. For details, see Section 9.1.1.3, « Editing Menu Entries during the Boot Procedure » (p. 159).

`/boot/grub/device.map`

This file translates device names from the GRUB and BIOS notation to Linux device names.

`/etc/grub.conf`

This file contains the commands, parameters and options the GRUB shell needs for installing the boot loader correctly.

`/etc/sysconfig/bootloader`

This file is read by the perl-bootloader library which is used when configuring the bootloader with YaST and every time a new kernel is installed. It contains configuration options (such as kernel parameters) that will be added by default to the bootloader configuration file.

GRUB can be controlled in various ways. Boot entries from an existing configuration can be selected from the graphical menu (splash screen). The configuration is loaded from the file `menu.lst`.

In GRUB, all boot parameters can be changed prior to booting. For example, errors made when editing the menu file can be corrected in this way. Boot commands can also be entered interactively at a kind of input prompt. For details, see Section 9.1.1.3, « Editing Menu Entries during the Boot Procedure » (p. 159). GRUB offers the possibility of determining the location of the kernel and the `initrd` prior to booting. In this way, you can even boot an installed operating system for which no entry exists in the boot loader configuration.

GRUB actually exists in two versions: as a boot loader and as a normal Linux program in `/usr/sbin/grub`. The latter is referred to as the *GRUB shell*. It provides an emulation of GRUB in the installed system and can be used to install GRUB or test new settings before applying them. The functionality to install GRUB as the boot loader on a hard disk or floppy disk is integrated in GRUB in the form of the command `setup`. This is available in the GRUB shell when Linux is loaded.

9.1.1 The File `/boot/grub/menu.lst`

The graphical splash screen with the boot menu is based on the GRUB configuration file `/boot/grub/menu.lst`, which contains all information about all partitions or operating systems that can be booted by the menu.

Every time the system is booted, GRUB loads the menu file from the file system. For this reason, GRUB does not need to be reinstalled after every change to the file. Use the YaST boot loader to modify the GRUB configuration as described in Section 9.2, « Configuring the Boot Loader with YaST » (p. 164).

The menu file contains commands. The syntax is very simple. Every line contains a command followed by optional parameters separated by spaces like in the shell. For historical reasons, some commands permit an `=` in front of the first parameter. Comments are introduced by a hash (`#`).

To identify the menu items in the menu overview, set a `title` for every entry. The text (including any spaces) following the keyword `title` is displayed as a selectable option in the menu. All commands up to the next `title` are executed when this menu item is selected.

The simplest case is the redirection to boot loaders of other operating systems. The command is `chainloader` and the argument is usually the boot block of another partition, in GRUB block notation. For example:

```
chainloader (hd0,3)+1
```

The device names in GRUB are explained in Section 9.1.1.1, « Naming Conventions for Hard Disks and Partitions » (p. 157). This example specifies the first block of the fourth partition of the first hard disk.

Use the command `kernel` to specify a kernel image. The first argument is the path to the kernel image in a partition. The other arguments are passed to the kernel on its command line.

If the kernel does not have built-in drivers for access to the root partition or a recent Linux system with advanced hotplug features is used, `initrd` must be specified with a separate GRUB command whose only argument is the path to the `initrd` file. Because the loading address of the `initrd` is written into the loaded kernel image, the command `initrd` must follow after the `kernel` command.

The command `root` simplifies the specification of kernel and `initrd` files. The only argument of `root` is a device or a partition. This device is used for all kernel,

`initrd`, or other file paths for which no device is explicitly specified until the next `root` command.

The `boot` command is implied at the end of every menu entry, so it does not need to be written into the menu file. However, if you use GRUB interactively for booting, you must enter the `boot` command at the end. The command itself has no arguments. It merely boots the loaded kernel image or the specified chain loader.

After writing all menu entries, define one of them as the `default` entry. Otherwise, the first one (entry 0) is used. You can also specify a time-out in seconds after which the default entry should boot. `timeout` and `default` usually precede the menu entries. An example file is described in Section 9.1.1.2, « An Example Menu File » (p. 158).

9.1.1.1 Naming Conventions for Hard Disks and Partitions

The naming convention GRUB uses for hard disks and partitions differ from that used for normal Linux devices. It more closely resembles the simple disk enumeration the BIOS does and the syntax is similar to that used in some BSD derivatives. In GRUB, the numbering of the partitions start with zero. This means that `(hd0, 0)` is the first partition of the first hard disk. On a common desktop machine with a hard disk connected as primary master, the corresponding Linux device name is `/dev/sda1`.

The four possible primary partitions are assigned the partition numbers 0 to 3. The logical partitions are numbered from 4:

<code>(hd0,0)</code>	first primary partition of the first hard disk
<code>(hd0,1)</code>	second primary partition
<code>(hd0,2)</code>	third primary partition
<code>(hd0,3)</code>	fourth primary partition (usually an extended partition)
<code>(hd0,4)</code>	first logical partition
<code>(hd0,5)</code>	second logical partition

Being dependent on BIOS devices, GRUB does not distinguish between PATA (IDE), SATA, SCSI, and hardware RAID devices. All hard disks recognized by the BIOS or other controllers are numbered according to the boot sequence preset in the BIOS.

Unfortunately, it is often not possible to map the Linux device names to BIOS device names exactly. It generates this mapping with the help of an algorithm and saves it to

the file `device.map`, which can be edited if necessary. Information about the file `device.map` is available in Section 9.1.2, « The File `device.map` » (p. 160).

A complete GRUB path consists of a device name written in parentheses and the path to the file in the file system in the specified partition. The path begins with a slash. For example, the bootable kernel could be specified as follows on a system with a single PATA (IDE) hard disk containing Linux in its first partition:

```
(hd0,0)/boot/vmlinuz
```

9.1.1.2 An Example Menu File

The following example shows the structure of a GRUB menu file. The example installation has a Linux boot partition under `/dev/sda5`, a root partition under `/dev/sda7` and a Windows installation under `/dev/sda1`.

```
gfxmenu (hd0,4)/boot/message❶
color white/blue black/light-gray❷
default 0❸
timeout 8❹

title linux❺
    root (hd0,4)
    kernel /boot/vmlinuz root=/dev/sda7 vga=791 resume=/dev/sda9
    initrd /boot/initrd

title windows❻
    rootnoverify (hd0,0)
    chainloader +1

title floppy❼
    rootnoverify (hd0,0)
    chainloader (fd0)+1

title failsafe❽
    root (hd0,4)
    kernel /boot/vmlinuz.shipped root=/dev/sda7 ide=nodma \
    apm=off acpi=off vga=normal nosmp maxcpus=0 3 noresume
    initrd /boot/initrd.shipped
```

The first block defines the configuration of the splash screen:

- ❶ The background image `message` is located in the `/boot` directory of the `/dev/sda5` partition.
- ❷ Color scheme: white (foreground), blue (background), black (selection) and light gray (background of the selection). The color scheme has no effect on the splash

screen, only on the customizable GRUB menu that you can access by exiting the splash screen with `Esc`.

- ❸ The first (0) menu entry `title linux` is booted by default.
- ❹ After eight seconds without any user input, GRUB automatically boots the default entry. To deactivate automatic boot, delete the `timeout` line. If you set `timeout 0`, GRUB boots the default entry immediately.

The second and largest block lists the various bootable operating systems. The sections for the individual operating systems are introduced by `title`.

- ❺ The first entry (`title linux`) is responsible for booting openSUSE. The kernel (`vmlinuz`) is located in the first logical partition (the boot partition) of the first hard disk. Kernel parameters, such as the root partition and VGA mode, are appended here. The root partition is specified according to the Linux naming convention (`/dev/sda7`) because this information is read by the kernel and has nothing to do with GRUB. The `initrd` is also located in the first logical partition of the first hard disk.
- ❻ The second entry is responsible for loading Windows. Windows is booted from the first partition of the first hard disk (`hd0, 0`). The command `chain-loader +1` causes GRUB to read and execute the first sector of the specified partition.
- ❼ The next entry enables booting from floppy disk without modifying the BIOS settings.
- ❽ The boot option `failsafe` starts Linux with a selection of kernel parameters that enables Linux to boot even on problematic systems.

The menu file can be changed whenever necessary. GRUB then uses the modified settings during the next boot. Edit the file permanently using YaST or an editor of your choice. Alternatively, make temporary changes interactively using the `edit` function of GRUB. See Section 9.1.1.3, « Editing Menu Entries during the Boot Procedure » (p. 159).

9.1.1.3 Editing Menu Entries during the Boot Procedure

In the graphical boot menu, select the operating system to boot with the arrow keys. If you select a Linux system, you can enter additional boot parameters at the boot prompt. To edit individual menu entries directly, press `Esc` to exit the splash screen and get to the GRUB text-based menu then press `E`. Changes made in this way only apply to the current boot and are not adopted permanently.

IMPORTANT : Keyboard Layout during the Boot Procedure

The US keyboard layout is the only one available when booting. See Figure « US Keyboard Layout » (↑Start-Up).

Editing menu entries facilitates the repair of a defective system that can no longer be booted, because the faulty configuration file of the boot loader can be circumvented by manually entering parameters. Manually entering parameters during the boot procedure is also useful for testing new settings without impairing the native system.

After activating the editing mode, use the arrow keys to select the menu entry of the configuration to edit. To make the configuration editable, press E again. In this way, edit incorrect partitions or path specifications before they have a negative effect on the boot process. Press Enter to exit the editing mode and return to the menu. Then press B to boot this entry. Further possible actions are displayed in the help text at the bottom.

To enter changed boot options permanently and pass them to the kernel, open the file `menu.lst` as the user `root` and append the respective kernel parameters to the existing line, separated by spaces:

```
title linux
    root (hd0,0)
    kernel /vmlinuz root=/dev/sda3 additional parameter
    initrd /initrd
```

GRUB automatically adopts the new parameters the next time the system is booted. Alternatively, this change can also be made with the YaST boot loader module. Append the new parameters to the existing line, separated by spaces.

9.1.2 The File `device.map`

The file `device.map` maps GRUB and BIOS device names to Linux device names. In a mixed system containing PATA (IDE) and SCSI hard disks, GRUB must try to determine the boot sequence by a special procedure, because GRUB may not have access to the BIOS information on the boot sequence. GRUB saves the result of this analysis in the file `/boot/grub/device.map`. Example `device.map` files for a system on which the boot sequence in the BIOS is set to PATA before SCSI could look as follows:

```
(fd0)    /dev/fd0
(hd0)    /dev/sda
(hd1)    /dev/sdb
```

or

```
(fd0) /dev/fd0
(hd0) /dev/disk-by-id/DISK1 ID
(hd1) /dev/disk-by-id/DISK2 ID
```

Because the order of PATA (IDE), SCSI and other hard disks depends on various factors and Linux is not able to identify the mapping, the sequence in the file `device.map` can be set manually. If you encounter problems when booting, check if the sequence in this file corresponds to the sequence in the BIOS and use the GRUB prompt to modify it temporarily, if necessary. After the Linux system has booted, the file `device.map` can be edited permanently with the YaST boot loader module or an editor of your choice.

After manually changing `device.map`, execute the following command to reinstall GRUB. This command causes the file `device.map` to be reloaded and the commands listed in `grub.conf` to be executed:

```
grub --batch < /etc/grub.conf
```

9.1.3 The File `/etc/grub.conf`

The third important GRUB configuration file after `menu.lst` and `device.map` is `/etc/grub.conf`. This file contains the commands, parameters and options the GRUB shell needs for installing the boot loader correctly:

```
setup --stage2=/boot/grub/stage2 --force-lba (hd0,1) (hd0,1)
quit
```

This command tells GRUB to automatically install the boot loader to the second partition on the first hard disk (`hd0,1`) using the boot images located on the same partition. The `--stage2=/boot/grub/stage2` parameter is needed to install the `stage2` image from a mounted file system. Some BIOSes have a faulty LBA support implementation, `--force-lba` provides a solution to ignore them.

9.1.4 The File `/etc/sysconfig/bootloader`

This configuration file is only used when configuring the bootloader with YaST and every time a new kernel is installed. It is evaluated by the `perl-bootloader` library which modifies the bootloader configuration file (for example `/boot/grub/menu.lst` for GRUB) accordingly. `/etc/sysconfig/bootloader` is not a GRUB specific configuration file - the values are applied to any bootloader installed on openSUSE.

REMARQUE : Bootloader Configuration after a Kernel Update

Every time a new kernel is installed, the perl bootloader writes a new boot-loader configuration file (for example `/boot/grub/menu.lst` for GRUB) using the defaults specified in `/etc/sysconfig/bootloader`. If you are using a customized set of kernel parameters, make sure to adjust the relevant defaults in `/etc/sysconfig/bootloader` according to your needs.

LOADER_TYPE

Specifies the bootloader installed on the system (e.g. GRUB or LILO). Do not modify—use YaST to change the bootloader as described in Procédure 9.6, « Changing the Boot Loader Type » (p. 169).

DEFAULT_VGA / FAILSAFE_VGA / XEN_VGA

Screen resolution and color depth of the framebuffer used during booting are configured with the kernel parameter `vga`. These values define which resolution and color depth to use for the default boot entry, the failsafe and the XEN entry. The following values are valid:

Tableau 9.1 *Screen Resolution and Color Depth Reference*

	640x480	800x600	1024x768	1280x1024	1600x1200
8bit	0x301	0x303	0x305	0x307	0x31C
15bit	0x310	0x313	0x316	0x319	0x31D
16bit	0x311	0x314	0x317	0x31A	0x31E
24bit	0x312	0x315	0x318	0x31B	0x31F

DEFAULT_APPEND / FAILSAFE_APPEND / XEN_KERNEL_APPEND

Kernel parameters (other than `vga`) that are automatically appended to the default, failsafe and XEN boot entries in the bootloader configuration file.

CYCLE_DETECTION / CYCLE_NEXT_ENTRY

Configure whether to use boot cycle detection and if so, which alternative entry from `/boot/grub/menu.lst` to boot in case of a reboot cycle (e.g. Failsafe). See `/usr/share/doc/packages/bootcycle/README` for detailed information.

9.1.5 Setting a Boot Password

Even before the operating system is booted, GRUB enables access to file systems. Users without root permissions can access files in your Linux system to which they have no access once the system is booted. To block this kind of access or to prevent users from booting certain operating systems, set a boot password.

IMPORTANT : Boot Password and Splash Screen

If you use a boot password for GRUB, the usual splash screen is not displayed.

As the user `root`, proceed as follows to set a boot password:

- 1 At the root prompt, encrypt the password using `grub-md5-crypt`:

```
# grub-md5-crypt
Password: ****
Retype password: ****
Encrypted: $1$lS2dv/$JOYcdxIn7CJk9xShzzJVw/
```

- 2 Paste the encrypted string into the global section of the file `menu.lst`:

```
gfxmenu (hd0,4)/message
color white/blue black/light-gray
default 0
timeout 8
password --md5 $1$lS2dv/$JOYcdxIn7CJk9xShzzJVw/
```

Now GRUB commands can only be executed at the boot prompt after pressing `P` and entering the password. However, users can still boot all operating systems from the boot menu.

- 3 To prevent one or several operating systems from being booted from the boot menu, add the entry `lock` to every section in `menu.lst` that should not be bootable without entering a password. For example:

```
title linux
    kernel (hd0,4)/vmlinuz root=/dev/sda7 vga=791
    initrd (hd0,4)/initrd
    lock
```

After rebooting the system and selecting the Linux entry from the boot menu, the following error message is displayed:

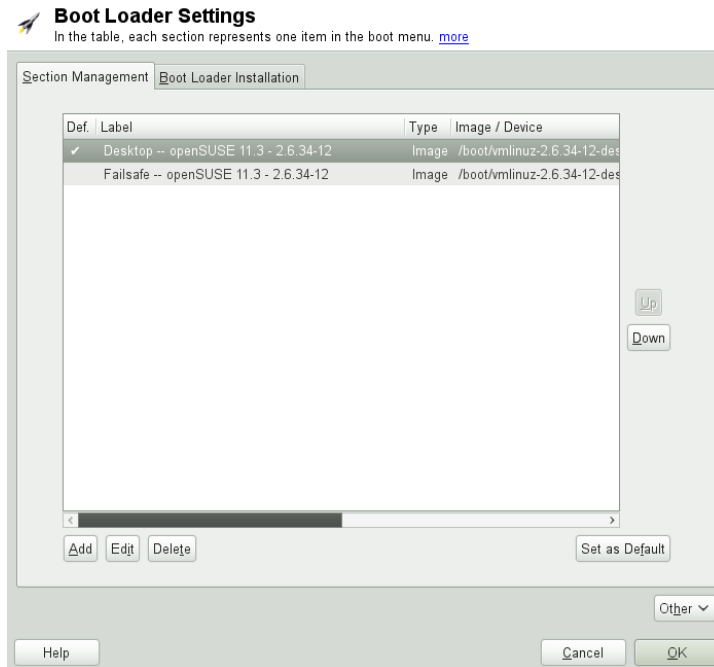
```
Error 32: Must be authenticated
```

Press Enter to enter the menu. Then press P to get a password prompt. After entering the password and pressing Enter, the selected operating system (Linux in this case) should boot.

9.2 Configuring the Boot Loader with YaST

The easiest way to configure the boot loader in your openSUSE system is to use the YaST module. In the YaST Control Center, select *System > Boot Loader*. As in Figure 9.1, « Boot Loader Settings » (p. 164), this shows the current boot loader configuration of your system and allows you to make changes.

Figure 9.1 *Boot Loader Settings*



Use the *Section Management* tab to edit, change and delete boot loader sections for the individual operating systems. To add an option, click *Add*. To change the value

of an existing option, select it with the mouse and click *Edit*. To remove an existing entry, select it and click *Delete*. If you are not familiar with boot loader options, read Section 9.1, « Booting with GRUB » (p. 154) first.

Use the *Boot Loader Installation* tab to view and change settings related to type, location and advanced loader settings.

Click *Other* to access advanced configuration options. The build-in editor lets you change the GRUB configuration files. For details, see Section 9.1, « Booting with GRUB » (p. 154). You can also delete the existing configuration and *Start from Scratch* or let YaST *Propose a New Configuration*. It is also possible to write the configuration to disk or reread the configuration from the disk. To restore the original Master Boot Record (MBR) that was saved during the installation, choose *Restore MBR of Hard Disk*.

9.2.1 Adjusting the Default Boot Entry

To change the system that is booted by default, proceed as follows:

Procédure 9.1 *Setting the Default System*

- 1 Open the *Section Management* tab.
- 2 Select the desired entry from the list.
- 3 Click *Set as Default*.
- 4 Click *OK* to activate these changes.

9.2.2 Modifying the Boot Loader Location

To modify the location of the boot loader, follow these steps:

Procédure 9.2 *Changing the Boot Loader Location*

- 1 Select the *Boot Loader Installation* tab and then choose one of the following options for *Boot Loader Location*:

Boot from Master Boot Record

This installs the boot loader in the MBR of the first disk (according to the boot sequence preset in the BIOS).

Boot from Root Partition

This installs the boot loader in the boot sector of the / partition (this is the default).

Boot from Boot Partition

This installs the boot loader in the boot sector of the /boot partition.

Boot from Extended Partition

This installs the boot loader in the extended partition container.

Custom Boot Partition

Use this option to specify the location of the boot loader manually.

- 2 Click *OK* to apply your changes.

9.2.3 Changing the Boot Loader Time-Out

The boot loader does not boot the default system immediately. During the time-out, you can select the system to boot or write some kernel parameters. To set the boot loader time-out, proceed as follows:

Procédure 9.3 *Changing the Boot Loader Time-Out*

- 1 Open the *Boot Loader Installation* tab.
- 2 Click *Boot Loader Options*.
- 3 Change the value of *Time-Out in Seconds* by typing in a new value and clicking the appropriate arrow key with your mouse, or by using the arrow keys on the keyboard.
- 4 Click *OK* twice to save the changes.

AVERTISSEMENT : Timeout of 0 Seconds

When setting the timeout to 0 seconds, you will not be able to access GRUB during boot time. When having set the default boot option to a non-Linux operation system at the same time, this effectively disables access to the Linux system.

9.2.4 Setting a Boot Password

Using this YaST module, you can also set a password to protect booting. This gives you an additional level of security.

Procédure 9.4 *Setting a Boot Loader Password*

- 1 Open the *Boot Loader Installation* tab.
- 2 Click *Boot Loader Options*.
- 3 Activate the *Protect Boot Loader with Password* option with a click and type in your *Password* twice.
- 4 Click *OK* twice to save the changes.

9.2.5 Adjusting the Disk Order

If your computer has more than one hard disk, you can specify the boot sequence of the disks to match the BIOS setup of the machine (see Section 9.1.2, « The File device.map » (p. 160)). To do so, proceed as follows:

Procédure 9.5 *Setting the Disk Order*

- 1 Open the *Boot Loader Installation* tab.
- 2 Click *Boot Loader Installation Details*.
- 3 If more than one disk is listed, select a disk and click *Up* or *Down* to reorder the displayed disks.
- 4 Click *OK* two times to save the changes.

9.2.6 Configuring Advanced Options

Advanced boot options can be configured via *Boot Loader Installation > Boot Loader Options*. Normally, it should not be necessary to change the default settings.

Set Active Flag in Partition Table for Boot Partition

Activates the partition that contains the boot loader. Some legacy operating systems (such as Windows 98) can only boot from an active partition.

Write Generic Boot Code to MBR

Replaces the current MBR with generic, operating system independent code.

Debugging Flag

Sets GRUB in debug mode where it displays messages to show disk activity.

Hide Boot Menu

Hides the boot menu and boots the default entry.

AVERTISSEMENT

When hiding the boot menu, you will not be able to access GRUB during boot time. When having set the default boot option to a non-Linux operation system at the same time, this effectively disables access to the Linux system.

Use Trusted GRUB

Starts the Trusted GRUB which supports trusted computing functionality.

Enable Acoustic Signals

Enables or disables acoustic signals in GRUB.

Graphical Menu File

Path to the graphics file used when displaying the boot screen.

Use Serial Console

If your machine is controlled via a serial console, activate this option and specify which COM port to use at which speed. See `info grub` or <http://www.gnu.org/software/grub/manual/grub.html#Serial-terminal>

9.2.7 Changing Boot Loader Type

Set the boot loader type in *Boot Loader Installation*. The default boot loader in openSUSE is GRUB. To use LILO or ELILO, proceed as follows:

AVERTISSEMENT : LILO is unsupported

Using LILO is not recommended—it is unsupported on openSUSE. Only use it in special cases.

Procédure 9.6 *Changing the Boot Loader Type*

- 1** Select the *Boot Loader Installation* tab.
- 2** For *Boot Loader*, select *LILO*.
- 3** In the dialog box that opens, select one of the following actions:

Propose New Configuration

Have YaST propose a new configuration.

Convert Current Configuration

Have YaST convert the current configuration. When converting the configuration, some settings may be lost.

Start New Configuration from Scratch

Write a custom configuration. This action is not available during the installation of openSUSE.

Read Configuration Saved on Disk

Load your own `/etc/lilo.conf`. This action is not available during the installation of openSUSE.

- 4** Click *OK* two times to save the changes.

During the conversion, the old GRUB configuration is saved to the disk. To use it, simply change the boot loader type back to GRUB and choose *Restore Configuration Saved before Conversion*. This action is available only on an installed system.

REMARQUE : Custom Boot Loader

To use a boot loader other than GRUB or LILO, select *Do Not Install Any Boot Loader*. Read the documentation of your boot loader carefully before choosing this option.

9.3 Uninstalling the Linux Boot Loader

YaST can be used to uninstall the Linux boot loader and restore the MBR to the state it had prior to the installation of Linux. During the installation, YaST automatically creates a backup copy of the original MBR and restores it upon request.

To uninstall GRUB, start YaST and click *System > Boot Loader* to start the boot loader module. Select *Other > Restore MBR of Hard Disk* and confirm with *Yes, Rewrite*.

9.4 Creating Boot CDs

If problems occur while booting your system using a boot manager or if the boot manager cannot be installed on your hard disk, it is also possible to create a bootable CD with all the necessary start-up files for Linux. This requires a CD writer be installed in your system.

Creating a bootable CD-ROM with GRUB merely requires a special form of *stage2* called *stage2_eltorito* and, optionally, a customized *menu.lst*. The classic files *stage1* and *stage2* are not required.

Procédure 9.7 *Creating Boot CDs*

- 1 Change into a directory in which to create the ISO image, for example: `cd /tmp`
- 2 Create a subdirectory for GRUB and change into the newly created `iso` directory:

```
mkdir -p iso/boot/grub && cd iso
```

- 3 Copy the kernel, the files *stage2_eltorito*, *initrd*, *menu.lst* and message to `iso/boot/`:

```
cp /boot/vmlinuz boot/  
cp /boot/initrd boot/  
cp /boot/message boot/  
cp /usr/lib/grub/stage2_eltorito boot/grub  
cp /boot/grub/menu.lst boot/grub
```

- 4 Replace the `root (hdx, y)` entries with `root (cd)` to point to the CD-ROM device. You may also need to adjust the paths to the message file, the kernel and the *initrd*—they need to point to `/boot/message`, `/boot/vmlinuz` and `/boot/initrd`, respectively. After having made the adjustments, *menu.lst* should look similar to the following example:

```
timeout 8  
default 0
```

```
gfxmenu (cd) /boot/message

title Linux
    root (cd)
    kernel /boot/vmlinuz root=/dev/sda5 vga=794 resume=/dev/sda1 \
    splash=verbose showopts
    initrd /boot/initrd
```

Use `splash=silent` instead of `splash=verbose` to prevent the boot messages from appearing during the boot procedure.

5 Create the ISO image with the following command:

```
genisoimage -R -b boot/grub/stage2_eltorito -no-emul-boot \
-boot-load-size 4 -boot-info-table -iso-level 2 -input-charset utf-8 \
-o grub.iso /tmp/iso
```

6 Write the resulting file `grub.iso` to a CD using your preferred utility. Do not burn the ISO image as a data file, but use the option for burning a CD image in your burning utility.

9.5 The Graphical SUSE Screen

The graphical SUSE screen is displayed on the first console if the option `vga=value` is used as a kernel parameter. If you install using YaST, this option is automatically activated in accordance with the selected resolution and the graphics card. There are three ways to disable the SUSE screen, if desired:

Disabling the SUSE Screen When Necessary

Enter the command `echo 0 >/proc/splash` on the command line to disable the graphical screen. To activate it again, enter `echo 1 >/proc/splash`.

Disabling the SUSE screen by default

Add the kernel parameter `splash=0` to your boot loader configuration. Chapitre 9, *The Boot Loader GRUB* (p. 153) provides more information about this. However, if you prefer the text mode (the default in earlier versions) set `vga=normal`.

Completely Disabling the SUSE Screen

Compile a new kernel and disable the option *Use splash screen instead of boot logo in framebuffer support*. Disabling framebuffer support in the kernel automatically disables the splash screen, as well.

AVERTISSEMENT : No Support

SUSE cannot provide any support for your system if you run it with a custom kernel.

9.6 Troubleshooting

This section lists some of the problems frequently encountered when booting with GRUB and a short description of possible solutions. Some of the problems are covered in articles in the Support Database at http://en.opensuse.org/Portal:Support_database. Use the search dialog to search for keywords like *GRUB*, *boot* and *boot loader*.

GRUB and XFS

XFS leaves no room for `stage1` in the partition boot block. Therefore, do not specify an XFS partition as the location of the boot loader. This problem can be solved by creating a separate boot partition that is not formatted with XFS.

GRUB Reports GRUB Geom Error

GRUB checks the geometry of connected hard disks when the system is booted. Sometimes, the BIOS returns inconsistent information and GRUB reports a GRUB Geom Error. In this case, update the BIOS.

GRUB also returns this error message if Linux was installed on an additional hard disk that is not registered in the BIOS. *stage1* of the boot loader is found and loaded correctly, but *stage2* is not found. This problem can be remedied by registering the new hard disk in the BIOS.

System Containing Several Hard Disks Does Not Boot

During the installation, YaST may have incorrectly determined the boot sequence of the hard disks. For example, GRUB may regard the PATA (IDE) disk as `hd0` and the SCSI disk as `hd1`, although the boot sequence in the BIOS is reversed (SCSI *before* PATA).

In this case, correct the hard disks during the boot process with the help of the GRUB command line. After the system has booted, edit `device.map` to apply the new mapping permanently. Then check the GRUB device names in the files `/boot/grub/menu.lst` and `/boot/grub/device.map` and reinstall the boot loader with the following command:

```
grub --batch < /etc/grub.conf
```

Booting Windows from the Second Hard Disk

Some operating systems, such as Windows, can only boot from the first hard disk. If such an operating system is installed on a hard disk other than the first hard disk, you can effect a logical change for the respective menu entry.

```
...
title windows
    map (hd0) (hd1)
    map (hd1) (hd0)
    chainloader (hd1,0)+1
...
```

In this example, Windows is started from the second hard disk. For this purpose, the logical order of the hard disks is changed with `map`. This change does not affect the logic within the GRUB menu file. Therefore, the second hard disk must be specified for `chainloader`.

9.7 For More Information

Extensive information about GRUB is available at <http://www.gnu.org/software/grub/>. Also refer to the `grub` info page. You can also search for the keyword « GRUB » in the Support Database at http://en.opensuse.org/Portal:Support_database to get information about special issues.

The Boot Loader GRUB2

This chapter describes how to configure GRUB2 (Grand Unified Bootloader), the boot loader used in openSUSE®. It is a successor of the traditional GRUB boot loader — now called « GRUB Legacy » — which is described in Chapitre 9, *The Boot Loader GRUB* (p. 153). GRUB2 has become the default boot loader in openSUSE® since version 12.2. A special YaST module is available for configuring major GRUB2 settings. If you are not familiar with the subject of booting in Linux, read the initial sections of Chapitre 9, *The Boot Loader GRUB* (p. 153) to acquire some background information. The boot procedure as a whole is outlined in Chapitre 7, *Booting a Linux System* (p. 127).

10.1 Main Differences from GRUB Legacy

- The configuration is stored in different files. The configuration syntax has changed.
- Partition numbers start at 1 (instead of 0 as in GRUB Legacy).
- More file-systems are supported.
- GRUB2 can directly read files stored on LVM or RAID devices.
- The user interface can be translated, including menu entry names.
- GRUB2 includes mechanism for loading modules to support specific features, such as filesystems etc.

- So called « Stages » were dropped and the images that make up GRUB2 were re-organized.

10.2 Configuration File Structure

The actual configuration of GRUB2 is based on the following files:

`/boot/grub2/grub.cfg`

This file contains all information about the GRUB2 menu items. It replaces `menu.lst` used in GRUB Legacy. `grub.cfg` is built by the `grub2-mkconfig` command, and normally is not edited manually.

`/etc/default/grub`

This file controls the user settings of GRUB2 and usually includes additional environmental settings such as backgrounds and themes.

Scripts under `/etc/grub.d/`

The scripts in this directory are read during execution of the `grub2-mkconfig` command. Their instructions are then incorporated into the main configuration file `/boot/grub/grub.cfg`.

`/etc/sysconfig/bootloader`

This file is read by the `perl-bootloader` library which is used when configuring the bootloader with YaST and every time a new kernel is installed. It contains configuration options (such as kernel parameters) that will be added by default to the bootloader configuration file.

GRUB2 can be controlled in various ways. Boot entries from an existing configuration can be selected from the graphical menu (splash screen). The configuration is loaded from the file `grub.cfg` which is compiled from other configuration files (see below). All GRUB2 configuration files are considered system files, and you need `root` privileges to edit them. Remember to run `grub2-mkconfig -o /boot/grub2/grub.cfg` after any change to GRUB2 configuration files.

10.2.1 The File `/boot/grub2/grub.cfg`

The graphical splash screen with the boot menu is based on the GRUB2 configuration file `/boot/grub2/grub.cfg`, which contains all information about all partitions or operating systems that can be booted by the menu.

Every time the system is booted, GRUB2 loads the menu file from the file system. For this reason, GRUB2 does not need to be re-installed after every change to the file. `grub.cfg` is automatically rebuilt with kernel installations or removals.

`grub.cfg` is compiled from the file `/etc/default/grub` and scripts found in the `/etc/grub.d/` directory with the `grub2-mkconfig -o /boot/grub2/grub.cfg` command. Therefore you should never edit the file manually. Instead, edit the related source files under `/etc/grub.d/` or use the YaST boot loader to modify the GRUB2 configuration as described in Section 10.3, « Configuring the Boot Loader with YaST » (p. 184).

10.2.2 The File `/etc/default/grub`

More general options of GRUB2 belong here, such as the time the menu is displayed, or the default OS to boot. To list all available options, see the output of the following command:

```
grep "export GRUB_DEFAULT" -A50 /usr/sbin/grub2-mkconfig | grep GRUB_
```

In addition to already defined variables, the user may introduce their own variables, and use them later in the scripts found in the `/etc/grub.d` directory.

After you edit `/etc/default/grub`, run `grub2-mkconfig -o /boot/grub2/grub.cfg` to update the main configuration file.

10.2.2.1 General Options

This section details some of the options commonly used in the `/etc/default/grub` file. For their complete list, see GNU GRUB manual [<http://www.gnu.org/software/grub/manual/grub.html#Simple-configuration>].

GRUB_DEFAULT

Sets the default menu entry that will be booted next time the computer is rebooted. It can be a numeric value, a complete menu entry quotation, or « saved ». A few examples follow:

`GRUB_DEFAULT=2` boots the third (counted from zero) boot menu entry.

`GRUB_DEFAULT="2>0"` boots the first submenu entry of the third top level menu entry.

`GRUB_DEFAULT="Example boot menu entry"` boots the menu entry whose title matches the quotation.

`GRUB_DEFAULT=saved` boots the entry specified by the `grub2-reboot` or `grub2-set-default` commands. While `grub2-reboot` sets the default boot entry for the next reboot only, `grub2-set-default` sets the default boot entry until changed.

`GRUB_SAVEDDEFAULT`

If set to `true`, it will automatically choose the last selected OS from the boot menu as the default boot entry on the next boot. For this to work, you also need to specify `GRUB_DEFAULT=saved`.

`GRUB_HIDDEN_TIMEOUT`

Waits the specified number of seconds for the user to press a key. During the period no menu is shown unless the user presses a key. If no key is pressed during the time specified, the control is passed to `GRUB_TIMEOUT`. `GRUB_HIDDEN_TIMEOUT=0` first checks whether `Maj` is pressed and shows the boot menu if yes, otherwise immediately boots the default menu entry. This is the default when only one bootable OS is identified by `GRUB2`.

`GRUB_HIDDEN_TIMEOUT_QUIET`

If `false` is specified, a countdown timer is displayed on a blank screen when the `GRUB_HIDDEN_TIMEOUT` feature is active.

`GRUB_TIMEOUT`

Time period in seconds the boot menu is displayed before automatically booting the default boot entry. If you press a key, the timeout is cancelled and `GRUB2` waits for you to make the selection manually. `GRUB_TIMEOUT=-1` will cause the menu to be displayed until you select the boot entry manually.

`GRUB_CMDLINE_LINUX`

Entries on this line are added to the end of the booting command line for both normal and recovery modes. It is used to pass options to the kernel.

`GRUB_CMDLINE_LINUX_DEFAULT`

Same as `GRUB_CMDLINE_LINUX` but the entries are passed and appended in the normal mode only.

GRUB_TERMINAL

Enables and specifies input/output terminal device. Can be `console` (PC BIOS and EFI consoles), `serial` (serial terminal), `ofconsole` (Open Firmware console), or the default `gfxterm` (graphics-mode output).

GRUB_GFXMODE

The resolution used for the `gfxterm` graphical terminal. Note that you can only use modes supported by your graphics card (VBE). The default is ‘auto’, which tries to select a preferred resolution. You can display the screen resolutions available to GRUB2 by typing `vbeinfo` in the GRUB2 command line. The command line is accessed by typing `c` when the GRUB2 boot menu screen is displayed.

You can also specify a color bit depth by appending it to the resolution setting, for example `GRUB_GFXMODE=1280x1024x24`.

ASTUCE

Setting the same resolution in GRUB2 and the operating system will slightly reduce the boot time.

GRUB_BACKGROUND

Set a background image for the `gfxterm` graphical terminal. The image must be a file readable by GRUB2 at boot time, and it must end with the `.png`, `.tga`, `.jpg`, or `.jpeg` suffix. If necessary, the image will be scaled to fit the screen.

10.2.3 Scripts in /etc/grub.d

The scripts in this directory are read during execution of the `grub2-mkconfig` command, and their instructions are incorporated into `/boot/grub2/grub.cfg`. The order of menu items in `grub.cfg` is determined by the order in which the files in this directory are run. Files with a leading numeral are executed first, beginning with the lowest number. `00_header` is run before `10_linux`, which would run before `40_custom`. If files with alphabetic names are present, they are executed after the numerically-named files. Only executable files generate output to `grub.cfg` during execution of `grub2-mkconfig`. By default all files in the `/etc/grub.d` directory are executable.

A list of default scripts follows.

00_header

Sets environmental variables such as system file locations, video settings, themes, and previously saved entries. It also imports preferences stored in the `/etc/default/grub`. Normally you do not need to make changes to this file.

10_linux

Identifies Linux kernels on the root device and creates relevant menu entries. This includes the associated recovery mode option if enabled. Only the latest kernel is displayed on the main menu page, with additional kernels included in a submenu.

30_os-prober

This script uses OS-prober to search for Linux and other operating systems and places the results in the GRUB2 menu. There are sections to identify specific operating systems, such as Linux, Windows, Hurd, or Mac OS X.

40_custom

A template for custom menu entries to be inserted into `grub.cfg`. The contents of this file, below the `exec tail -n +3 $0` line and the default comments, are imported directly into `grub.cfg` without any changes.

90_persistent

This is a special script which copies a corresponding part of the `grub.cfg` file and outputs it back unchanged. This way you can modify that part of `grub.cfg` directly and the change survives the execution of `grub2-mkconfig`.

10.2.4 The File `/etc/sysconfig/bootloader`

This configuration file is only used when configuring the bootloader with YaST and every time a new kernel is installed. It is evaluated by the perl-bootloader library which modifies the bootloader configuration file (for example `/boot/grub2/grub.cfg` for GRUB2) accordingly. `/etc/sysconfig/bootloader` is not a GRUB2 specific configuration file - the values are applied to any bootloader installed on openSUSE.

REMARQUE : Bootloader Configuration after a Kernel Update

Every time a new kernel is installed, the perl bootloader writes a new bootloader configuration file using the defaults specified in `/etc/sysconfig/`

`bootloader`. If you are using a customized set of kernel parameters, make sure to adjust the relevant defaults in `/etc/sysconfig/bootloader` according to your needs.

LOADER_TYPE

Specifies the bootloader installed on the system (e.g. GRUB2, GRUB, or LILO). Do not modify—use YaST to change the bootloader as described in Section 10.3.5, « Changing Boot Loader Type » (p. 188).

LOADER_LOCATION

Specifies the location of the bootloader. Do not modify—use YaST to change the bootloader as described in Section 10.3.2, « Modifying the Boot Loader Location » (p. 185).

DEFAULT_VGA / FAILSAFE_VGA / XEN_VGA

Screen resolution and color depth of the framebuffer used during booting are configured with the kernel parameter `vga`. These values define which resolution and color depth to use for the default boot entry, the failsafe and the XEN entry. The following values are valid:

Tableau 10.1 *Screen Resolution and Color Depth Reference*

	640x480	800x600	1024x768	1280x1024	1600x1200
8bit	0x301	0x303	0x305	0x307	0x31C
15bit	0x310	0x313	0x316	0x319	0x31D
16bit	0x311	0x314	0x317	0x31A	0x31E
24bit	0x312	0x315	0x318	0x31B	0x31F

DEFAULT_APPEND / FAILSAFE_APPEND / XEN_KERNEL_APPEND

Kernel parameters (other than `vga`) that are automatically appended to the default, failsafe and XEN boot entries in the bootloader configuration file.

CYCLE_DETECTION / CYCLE_NEXT_ENTRY

Configure whether to use boot cycle detection and if so, which alternative boot entry from to boot in case of a reboot cycle (e.g. Failsafe). See `/usr/share/doc/packages/bootcycle/README` for detailed information.

10.2.5 Mapping between BIOS drives and Linux devices

In GRUB Legacy, the `device.map` configuration file was used to derive Linux device names from BIOS drive numbers. The map between BIOS drives and Linux devices cannot always be guessed correctly: for example, GRUB will get the order wrong if you exchange the boot sequence between IDE and SCSI in your BIOS.

GRUB2 avoids this problem by using device ID strings (UUIDs) or file system labels when generating `grub.cfg`. GRUB2 utilities create a temporary device map on the fly, which is usually sufficient, particularly in the case of single-disk systems.

However, if you need to override the GRUB2's automatic device mapping mechanism, create your custom mapping file `/boot/grub2/device.map` similarly as in GRUB Legacy (see Section 9.1.2, « The File `device.map` » (p. 160) for more details). Please note that GRUB2 partition number start at 1 and not at 0 as in GRUB Legacy.

10.2.6 Editing Menu Entries during the Boot Procedure

Editing menu entries facilitates the repair of a defective system that can no longer be booted, because the faulty configuration file of the boot loader can be circumvented by manually entering parameters. Manually entering parameters during the boot procedure is also useful for testing new settings without impairing the native system.

In the graphical boot menu, select the operating system to boot with the arrow keys. If you select a Linux system, you can enter additional boot parameters at the boot prompt. To edit individual menu entries directly, press `Esc` to exit the splash screen and get to the GRUB2 text-based menu then press `E`. Changes made in this way only apply to the current boot and are not adopted permanently.

REMARQUE : Password-protected Bootloader

If you protected your bootloader with a password as described in Section 10.2.7, « Setting a Boot Password » (p. 183), you need to first enter the specified username and password to « unlock » the bootloader.

IMPORTANT : Keyboard Layout during the Boot Procedure

The US keyboard layout is the only one available when booting. See Figure « US Keyboard Layout » (↑Start-Up).

Configuration options and commands related to the selected boot entry appears in a simple text editor interface. Use arrows to move in the text and modify the menu entry as desired. The TAB key suggests possible completions, while ESC discards all changes and returns to the GRUB2 menu. Once you finish editing the menu-entry, press F10 to boot it.

10.2.7 Setting a Boot Password

Even before the operating system is booted, GRUB2 enables access to file systems. Users without root permissions can access files in your Linux system to which they have no access once the system is booted. To block this kind of access or to prevent users from booting certain operating systems, set a boot password.

IMPORTANT : Boot Password and Splash Screen

If you use a boot password for GRUB2, the usual splash screen is not displayed.

As the user `root`, proceed as follows to set a boot password:

- 1 At the root prompt, encrypt the password using `grub2-mkpasswd-pbkdf2`:

```
# grub2-mkpasswd-pbkdf2
Password: ****
Reenter password: ****
PBKDF2 hash of your password is
grub.pbkdf2.sha512.10000.9CA4611006FE96BC77A...
```

- 2 Paste the encrypted long string into the file `/etc/grub.d/40_custom` together with the `set superusers` command. Remember to keep the commented lines at the beginning:

```
set superusers="root"
password_pbkdf2 root grub.pbkdf2.sha512.10000.9CA4611006FE96BC77A...
```

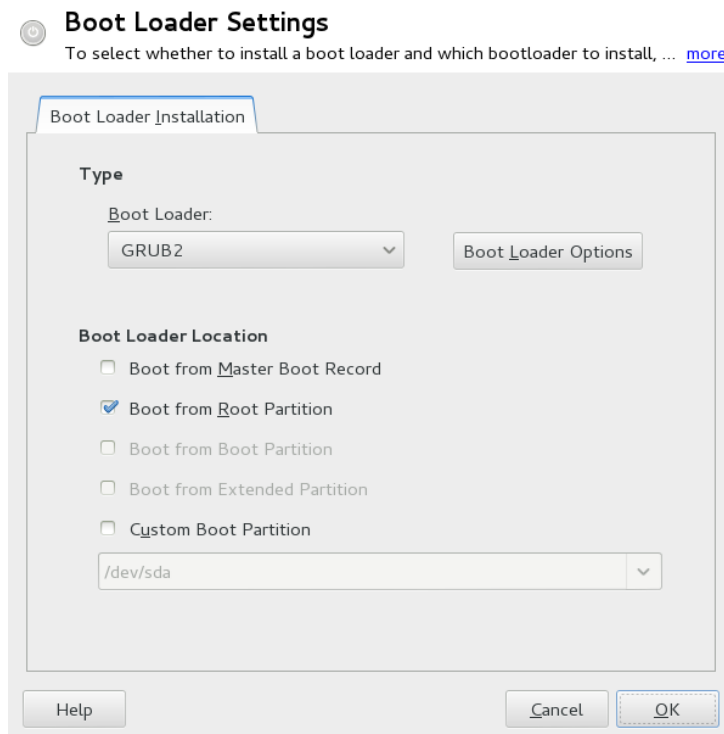
- 3 Run `grub2-mkconfig -o /boot/grub2/grub.cfg` to import the changes into the main configuration file.

After you reboot, you will be prompted for username and password when trying to boot any menu entry. Enter `root` and the password you typed during the `grub2-mkpasswd-pbkdf2` command. If the credentials are correct, the system will boot the selected boot entry.

10.3 Configuring the Boot Loader with YaST

The easiest way to configure general options of the boot loader in your openSUSE system is to use the YaST module. In the YaST Control Center, select *System > Boot Loader*. As in Figure 9.1, « Boot Loader Settings » (p. 164), this shows the current boot loader configuration of your system and allows you to make changes.

Figure 10.1 *Boot Loader Settings*



Use the *Boot Loader Installation* tab to view and change settings related to type, location and advanced loader settings. To use the GRUB2 boot loader, make sure it is selected from the list of available boot loaders.

10.3.1 Adjusting the Default Boot Entry

To change the system that is booted by default, proceed as follows:

Procédure 10.1 *Setting the Default System*

- 1 Click *Boot Loader Options* and open the *Default Boot Section* list.
- 2 Select the desired entry from the list. Note that the « > » sign in the boot entry name delimits boot section and its subsection.
- 3 Click *OK* to activate these changes.

10.3.2 Modifying the Boot Loader Location

To modify the location of the boot loader, follow these steps:

Procédure 10.2 *Changing the Boot Loader Location*

- 1 Select the *Boot Loader Installation* tab and then choose one of the following options for *Boot Loader Location*:

Boot from Master Boot Record

This installs the boot loader in the MBR of the first disk (according to the boot sequence preset in the BIOS).

Boot from Root Partition

This installs the boot loader in the boot sector of the / partition (this is the default).

Boot from Boot Partition

This installs the boot loader in the boot sector of the /boot partition.

Boot from Extended Partition

This installs the boot loader in the extended partition container.

Custom Boot Partition

Use this option to specify the location of the boot loader manually.

- 2 Click *OK* to apply your changes.

10.3.3 Changing the Boot Loader Time-Out

The boot loader does not boot the default system immediately. During the time-out, you can select the system to boot or write some kernel parameters. To set the boot loader time-out, proceed as follows:

Procédure 10.3 *Changing the Boot Loader Time-Out*

- 1 Open the *Boot Loader Installation* tab.
- 2 Click *Boot Loader Options*.
- 3 Change the value of *Time-Out in Seconds* by typing in a new value and clicking the appropriate arrow key with your mouse, or by using the arrow keys on the keyboard.
- 4 Click *OK* twice to save the changes.

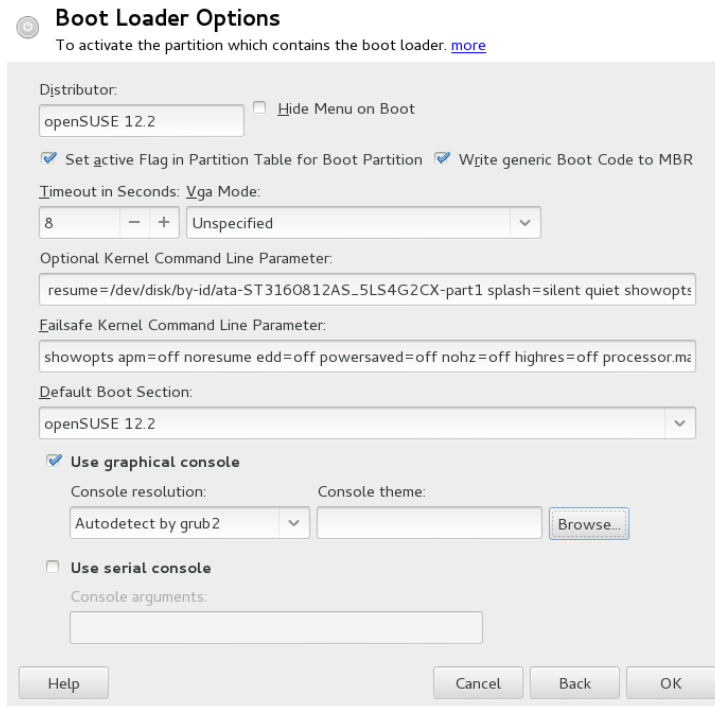
AVERTISSEMENT : Timeout of 0 Seconds

When setting the timeout to 0 seconds, you will not be able to access GRUB2 during boot time. When having set the default boot option to a non-Linux operation system at the same time, this effectively disables access to the Linux system.

10.3.4 Configuring Advanced Options

Advanced boot options can be configured via *Boot Loader Installation > Boot Loader Options*. Normally, it should not be necessary to change the default settings.

Figure 10.2 *Boot Loader Options*



Boot Loader Options
To activate the partition which contains the boot loader: [more](#)

Distributor:
openSUSE 12.2 ☐ Hide Menu on Boot

☒ Set active Flag in Partition Table for Boot Partition ☒ Write generic Boot Code to MBR

Timeout in Seconds: Vga Mode:
8 - + Unspecified

Optional Kernel Command Line Parameter:
resume=/dev/disk/by-id/ata-ST3160812AS_5LS4G2CX-part1 splash=silent quiet showopts

Failsafe Kernel Command Line Parameter:
showopts apm=off noresume edd=off powersaved=off nohz=off highres=off processor.m

Default Boot Section:
openSUSE 12.2

☒ Use graphical console
Console resolution: Autodetect by grub2 Console theme: Browse...

☐ Use serial console
Console arguments:

Help Cancel Back OK

Set Active Flag in Partition Table for Boot Partition

Activates the partition that contains the boot loader. Some legacy operating systems (such as Windows 98) can only boot from an active partition.

Write Generic Boot Code to MBR

Replaces the current MBR with generic, operating system independent code.

Hide Menu on Boot

Hides the boot menu and boots the default entry.

Use graphical console

When checked, the boot menu appears on a graphical splash screen rather than in a text mode. The resolution of the boot screen can be then set from the *Console*

resolution list, and also graphical theme definition file can be specified with the *Console theme* file-chooser.

Use Serial Console

If your machine is controlled via a serial console, activate this option and specify which COM port to use at which speed. See `info grub` or <http://www.gnu.org/software/grub/manual/grub.html#Serial-terminal>

10.3.5 Changing Boot Loader Type

Set the boot loader type in *Boot Loader Installation*. The default boot loader in openSUSE is GRUB2. To use GRUB, LILO or ELILO, proceed as follows:

AVERTISSEMENT : LILO is unsupported

Using LILO is not recommended—it is unsupported on openSUSE. Only use it in special cases.

Procédure 10.4 *Changing the Boot Loader Type*

- 1 Select the *Boot Loader Installation* tab.
- 2 For *Boot Loader*, select *GRUB* or *LILO*.
- 3 In the dialog box that opens, select one of the following actions:

Propose New Configuration

Have YaST propose a new configuration.

Convert Current Configuration

Have YaST convert the current configuration. When converting the configuration, some settings may be lost.

Start New Configuration from Scratch

Write a custom configuration. This action is not available during the installation of openSUSE.

Read Configuration Saved on Disk

Load your own bootloader configuration file. This action is not available during the installation of openSUSE.

4 Click *OK* two times to save the changes.

During the conversion, the old GRUB2 configuration is saved to the disk. To use it, simply change the boot loader type back to GRUB2 and choose *Restore Configuration Saved before Conversion*. This action is available only on an installed system.

REMARQUE : Custom Boot Loader

To use a boot loader other than GRUB2, GRUB, or LILO, select *Do Not Install Any Boot Loader*. Read the documentation of your boot loader carefully before choosing this option.

10.4 For More Information

Extensive information about GRUB2 is available at <http://www.gnu.org/software/grub/>. Also refer to the `grub` info page. You can also search for the keyword « GRUB2 » in the Support Database at http://en.opensuse.org/Portal:Support_database to get information about special issues.

Special System Features

This chapter starts with information about various software packages, the virtual consoles and the keyboard layout. We talk about software components like `bash`, `cron` and `logrotate`, because they were changed or enhanced during the last release cycles. Even if they are small or considered of minor importance, users may want to change their default behavior, because these components are often closely coupled with the system. The chapter concludes with a section about language and country-specific settings (I18N and L10N).

11.1 Information about Special Software Packages

The programs `bash`, `cron`, `logrotate`, `locate`, `ulimit` and `free` are very important for system administrators and many users. Man pages and info pages are two useful sources of information about commands, but both are not always available. GNU Emacs is a popular and very configurable text editor.

11.1.1 The `bash` Package and `/etc/profile`

Bash is the default system shell. When used as a login shell, it reads several initialization files. Bash processes them in the order they appear in this list:

1. `/etc/profile`

2. `~/ .profile`

3. `/etc/bash.bashrc`

4. `~/ .bashrc`

Make custom settings in `~/ .profile` or `~/ .bashrc`. To ensure the correct processing of these files, it is necessary to copy the basic settings from `/etc/skel/.profile` or `/etc/skel/.bashrc` into the home directory of the user. It is recommended to copy the settings from `/etc/skel` after an update. Execute the following shell commands to prevent the loss of personal adjustments:

```
mv ~/.bashrc ~/.bashrc.old
cp /etc/skel/.bashrc ~/.bashrc
mv ~/.profile ~/.profile.old
cp /etc/skel/.profile ~/.profile
```

Then copy personal adjustments back from the `*.old` files.

11.1.2 The cron Package

If you want to run commands regularly and automatically in the background at predefined times, cron is the tool to use. cron is driven by specially formatted time tables. Some of them come with the system and users can write their own tables if needed.

The cron tables are located in `/var/spool/cron/tabs`. `/etc/crontab` serves as a systemwide cron table. Enter the username to run the command directly after the time table and before the command. In Example 11.1, « Entry in `/etc/crontab` » (p. 192), `root` is entered. Package-specific tables, located in `/etc/cron.d`, have the same format. See the cron man page (`man cron`).

Example 11.1 *Entry in `/etc/crontab`*

```
1-59/5 * * * * root test -x /usr/sbin/atrun && /usr/sbin/atrun
```

You cannot edit `/etc/crontab` by calling the command `crontab -e`. This file must be loaded directly into an editor, then modified and saved.

A number of packages install shell scripts to the directories `/etc/cron.hourly`, `/etc/cron.daily`, `/etc/cron.weekly` and `/etc/cron.monthly`, whose execution is controlled by `/usr/lib/cron/run-crons`. `/usr/lib/cron/run-crons` is run every 15 minutes from the main table (`/etc/crontab`). This guarantees that processes that may have been neglected can be run at the proper time.

To run the `hourly`, `daily` or other periodic maintenance scripts at custom times, remove the time stamp files regularly using `/etc/crontab` entries (see Example 11.2, « `/etc/crontab`: Remove Time Stamp Files » (p. 193), which removes the `hourly` one before every full hour, the `daily` one once a day at 2:14 a.m., etc.).

Example 11.2 */etc/crontab: Remove Time Stamp Files*

```
59 * * * * root rm -f /var/spool/cron/lastrun/cron.hourly
14 2 * * * root rm -f /var/spool/cron/lastrun/cron.daily
29 2 * * 6 root rm -f /var/spool/cron/lastrun/cron.weekly
44 2 1 * * root rm -f /var/spool/cron/lastrun/cron.monthly
```

Or you can set `DAILY_TIME` in `/etc/sysconfig/cron` to the time at which `cron.daily` should start. The setting of `MAX_NOT_RUN` ensures that the daily tasks get triggered to run, even if the user did not turn on the computer at the specified `DAILY_TIME` for a longer period of time. The maximum value of `MAX_NOT_RUN` is 14 days.

The daily system maintenance jobs are distributed to various scripts for reasons of clarity. They are contained in the package `aaa_base`. `/etc/cron.daily` contains, for example, the components `suse.de-backup-rpmdb`, `suse.de-clean-tmp` or `suse.de-cron-local`.

11.1.3 Log Files: Package logrotate

There are a number of system services (*daemons*) that, along with the kernel itself, regularly record the system status and specific events onto log files. This way, the administrator can regularly check the status of the system at a certain point in time, recognize errors or faulty functions and troubleshoot them with pinpoint precision. These log files are normally stored in `/var/log` as specified by FHS and grow on a daily basis. The `logrotate` package helps control the growth of these files.

Configure `logrotate` with the file `/etc/logrotate.conf`. In particular, the `include` specification primarily configures the additional files to read. Programs that produce log files install individual configuration files in `/etc/logrotate.d`. For example, such files ship with the packages `apache2` (`/etc/logrotate.d/apache2`) and `syslogd` (`/etc/logrotate.d/syslog`).

Example 11.3 *Example for /etc/logrotate.conf*

```
# see "man logrotate" for details
# rotate log files weekly
```

```
weekly

# keep 4 weeks worth of backlogs
rotate 4

# create new (empty) log files after rotating old ones
create

# uncomment this if you want your log files compressed
#compress

# RPM packages drop log rotation information into this directory
include /etc/logrotate.d

# no packages own lastlog or wtmp - we'll rotate them here
#/var/log/wtmp {
#   monthly
#   create 0664 root utmp
#   rotate 1
#}

# system-specific logs may be also be configured here.
```

logrotate is controlled through cron and is called daily by `/etc/cron.daily/logrotate`.

IMPORTANT

The `create` option reads all settings made by the administrator in `/etc/permissions*`. Ensure that no conflicts arise from any personal modifications.

11.1.4 The locate Command

`locate`, a command for quickly finding files, is not included in the standard scope of installed software. If desired, install the package `findutils-locate`. The `updatedb` process is started automatically every night or about 15 minutes after booting the system.

11.1.5 The ulimit Command

With the `ulimit` (*user limits*) command, it is possible to set limits for the use of system resources and to have these displayed. `ulimit` is especially useful for limiting available memory for applications. With this, an application can be prevented

from co-opting too much of the system resources and slowing or even hanging up the operating system.

`ulimit` can be used with various options. To limit memory usage, use the options listed in Tableau 11.1, « `ulimit`: Setting Resources for the User » (p. 195).

Tableau 11.1 *ulimit: Setting Resources for the User*

-m	The maximum resident set size
-v	The maximum amount of virtual memory available to the shell
-s	The maximum size of the stack
-c	The maximum size of core files created
-a	All current limits are reported

Systemwide entries can be made in `/etc/profile`. There, enable creation of core files (needed by programmers for *debugging*). A normal user cannot increase the values specified in `/etc/profile` by the system administrator, but can make special entries in `~/ .bashrc`.

Exemple 11.4 *ulimit: Settings in `~/ .bashrc`*

```
# Limits maximum resident set size (physical memory):
ulimit -m 98304

# Limits of virtual memory:
ulimit -v 98304
```

Memory allocations must be specified in KB. For more detailed information, see `man bash`.

IMPORTANT

Not all shells support `ulimit` directives. PAM (for instance, `pam_limits`) offers comprehensive adjustment possibilities if you depend on encompassing settings for these restrictions.

11.1.6 The free Command

The `free` command displays the total amount of free and used physical memory and swap space in the system, as well as the buffers and cache consumed by the kernel. The concept of *available RAM* dates back to before the days of unified memory management. The slogan *free memory is bad memory* applies well to Linux. As a result, Linux has always made the effort to balance out caches without actually allowing free or unused memory.

Basically, the kernel does not have direct knowledge of any applications or user data. Instead, it manages applications and user data in a *page cache*. If memory runs short, parts of it are written to the swap partition or to files, from which they can initially be read with the help of the `mmap` command (see `man mmap`).

The kernel also contains other caches, such as the *slab cache*, where the caches used for network access are stored. This may explain the differences between the counters in `/proc/meminfo`. Most, but not all, of them can be accessed via `/proc/slabinfo`.

However, if your goal is to find out how much RAM is currently being used, find this information in `/proc/meminfo`.

11.1.7 Man Pages and Info Pages

For some GNU applications (such as `tar`), the man pages are no longer maintained. For these commands, use the `--help` option to get a quick overview of the info pages, which provide more in-depth instructions. Info is GNU's hypertext system. Read an introduction to this system by entering `info info`. Info pages can be viewed with Emacs by entering `emacs -f info` or directly in a console with `info`. You can also use `tkinfo`, `xinfo` or the help system to view info pages.

11.1.8 Selecting Man Pages Using the man Command

To read a man page enter `man man_page`. If a man page with the same name exists in different sections, they will all be listed with the corresponding section numbers. Select the one to display. If you don't enter a section number within a few seconds, the first man page will be displayed.

If you want to change this to the default system behavior, set `MAN_POSIXLY_CORRECT=1` in a shell initialization file such as `~/.bashrc`.

11.1.9 Settings for GNU Emacs

GNU Emacs is a complex work environment. The following sections cover the configuration files processed when GNU Emacs is started. More information is available at <http://www.gnu.org/software/emacs/>.

On start-up, Emacs reads several files containing the settings of the user, system administrator and distributor for customization or preconfiguration. The initialization file `~/.emacs` is installed to the home directories of the individual users from `/etc/skel/.emacs`, in turn, reads the file `/etc/skel/.gnu-emacs`. To customize the program, copy `.gnu-emacs` to the home directory (with `cp /etc/skel/.gnu-emacs ~/.gnu-emacs`) and make the desired settings there.

`.gnu-emacs` defines the file `~/.gnu-emacs-custom` as `custom-file`. If users make settings with the `customize` options in Emacs, the settings are saved to `~/.gnu-emacs-custom`.

With openSUSE, the `emacs` package installs the file `site-start.el` in the directory `/usr/share/emacs/site-lisp`. The file `site-start.el` is loaded before the initialization file `~/.emacs`. Among other things, `site-start.el` ensures that special configuration files distributed with Emacs add-on packages, such as `psgml`, are loaded automatically. Configuration files of this type are located in `/usr/share/emacs/site-lisp`, too, and always begin with `suse-start-`. The local system administrator can specify systemwide settings in `default.el`.

More information about these files is available in the Emacs info file under *Init File*: <info:/emacs/InitFile>. Information about how to disable the loading of these files (if necessary) is also provided at this location.

The components of Emacs are divided into several packages:

- The base package `emacs`.
- `emacs-x11` (usually installed): the program *with* X11 support.
- `emacs-nox`: the program *without* X11 support.
- `emacs-info`: online documentation in info format.

- `emacs-el`: the uncompiled library files in Emacs Lisp. These are not required at runtime.
- Numerous add-on packages can be installed if needed: `emacs-auctex` (LaTeX), `psgml` (SGML and XML), `gnuserv` (client and server operation) and others.

11.2 Virtual Consoles

Linux is a multiuser and multitasking system. The advantages of these features can be appreciated even on a stand-alone PC system. In text mode, there are six virtual consoles available. Switch between them using `Alt + F1` through `Alt + F6`. The seventh console is reserved for X and the tenth console shows kernel messages. More or fewer consoles can be assigned by modifying the file `/etc/inittab`.

To switch to a console from X without shutting it down, use `Ctrl + Alt + F1` to `Ctrl + Alt + F6`. To return to X, press `Alt + F7`.

11.3 Keyboard Mapping

To standardize the keyboard mapping of programs, changes were made to the following files:

```
/etc/inputrc
/etc/X11/Xmodmap
/etc/skel/.emacs
/etc/skel/.gnu-emacs
/etc/skel/.vimrc
/etc/csh.cshrc
/etc/termcap
/usr/share/terminfo/x/xterm
/usr/share/X11/app-defaults/XTerm
/usr/share/emacs/VERSION/site-lisp/term/*.el
```

These changes only affect applications that use `terminfo` entries or whose configuration files are changed directly (`vi`, `emacs`, etc.). Applications not shipped with the system should be adapted to these defaults.

Under X, the compose key (multikey) can be enabled as explained in `/etc/X11/Xmodmap`.

Further settings are possible using the X Keyboard Extension (XKB). This extension is also used by the desktop environments GNOME (gswitchit) and KDE (kxkb).

ASTUCE : For More Information

Information about XKB is available in the documents listed in `/usr/share/doc/packages/xkeyboard-config` (part of the `xkeyboard-config` package).

11.4 Language and Country-Specific Settings

The system is, to a very large extent, internationalized and can be modified for local needs. Internationalization (*I18N*) allows specific localizations (*L10N*). The abbreviations *I18N* and *L10N* are derived from the first and last letters of the words *and*, *in* *between*, the number of letters omitted.

Settings are made with `LC_` variables defined in the file `/etc/sysconfig/language`. This refers not only to *native language support*, but also to the categories *Messages* (Language), *Character Set*, *Sort Order*, *Time and Date*, *Numbers* and *Money*. Each of these categories can be defined directly with its own variable or indirectly with a master variable in the file `language` (see the `locale` man page).

`RC_LC_MESSAGES`, `RC_LC_CTYPE`, `RC_LC_COLLATE`, `RC_LC_TIME`,
`RC_LC_NUMERIC`, `RC_LC_MONETARY`

These variables are passed to the shell without the `RC_` prefix and represent the listed categories. The shell profiles concerned are listed below. The current setting can be shown with the command `locale`.

`RC_LC_ALL`

This variable, if set, overwrites the values of the variables already mentioned.

`RC_LANG`

If none of the previous variables are set, this is the fallback. By default, only `RC_LANG` is set. This makes it easier for users to enter their own values.

`ROOT_USES_LANG`

A yes or no variable. If set to `no`, `root` always works in the POSIX environment.

The variables can be set with the YaST sysconfig editor. The value of such a variable contains the language code, country code, encoding and modifier. The individual components are connected by special characters:

```
LANG=<language>[_<COUNTRY>].<Encoding>[@<Modifier>]]
```

11.4.1 Some Examples

You should always set the language and country codes together. Language settings follow the standard ISO 639 available at <http://www.evertype.com/standards/iso639/iso639-en.html> and <http://www.loc.gov/standards/iso639-2/>. Country codes are listed in ISO 3166 available at http://www.din.de/gremien/nas/nabd/iso3166ma/codlstp1/en_listp1.html.

It only makes sense to set values for which usable description files can be found in `/usr/lib/locale`. Additional description files can be created from the files in `/usr/share/i18n` using the command `localedef`. The description files are part of the `glibc-i18ndata` package. A description file for `en_US.UTF-8` (for English and United States) can be created with:

```
localedef -i en_US -f UTF-8 en_US.UTF-8
```

```
LANG=en_US.UTF-8
```

This is the default setting if American English is selected during installation. If you selected another language, that language is enabled but still with UTF-8 as the character encoding.

```
LANG=en_US.ISO-8859-1
```

This sets the language to English, country to United States and the character set to ISO-8859-1. This character set does not support the Euro sign, but it can be useful sometimes for programs that have not been updated to support UTF-8. The string defining the charset (`ISO-8859-1` in this case) is then evaluated by programs like Emacs.

```
LANG=en_IE@euro
```

The above example explicitly includes the Euro sign in a language setting. This setting is basically obsolete now, as UTF-8 also covers the Euro symbol. It is only useful if an application supports ISO-8859-15 and not UTF-8.

In former releases, it was necessary to run `SuSEconfig` after doing any changes to `/etc/sysconfig/language`. `SuSEconfig` then wrote the changes to `/etc/SuSEconfig/profile` and `/etc/SuSEconfig/csh.login`. Upon login, these files were read by `/etc/profile` (for the `Bash`) or by `/etc/csh.login` (for the `tcsh`) .

In recent releases, `/etc/SuSEconfig/profile` has been replaced with `/etc/profile.d/lang.sh`, and `/etc/SuSEconfig/csh.login` with `/etc/profile.d/lang.csh`. But if they exist, both legacy file are still read upon login.

The process chain is now as follows:

- For the `Bash`: `/etc/profile` reads `/etc/profile.d/lang.sh` which, in turn, analyzes `/etc/sysconfig/language`.
- For `tcsh`: At login, `/etc/csh.login` reads `/etc/profile.d/lang.csh` which, in turn, analyzes `/etc/sysconfig/language`.

This ensures that any changes to `/etc/sysconfig/language` are available at the next login to the respective shell, without having to run `SuSEconfig` first.

Users can override the system defaults by editing their `~/.bashrc` accordingly. For instance, if you do not want to use the systemwide `en_US` for program messages, include `LC_MESSAGES=es_ES` so that messages are displayed in Spanish instead.

11.4.2 Locale Settings in `~/.i18n`

If you are not satisfied with locale system defaults, change the settings in `~/.i18n` according to the `Bash` scripting syntax. Entries in `~/.i18n` override system defaults from `/etc/sysconfig/language`. Use the same variable names but without the `RC_` namespace prefixes. For example, use `LANG` instead of `RC_LANG`:

```
LANG=cs_CZ.UTF-8
LC_COLLATE=C
```

11.4.3 Settings for Language Support

Files in the category *Messages* are, as a rule, only stored in the corresponding language directory (like `en`) to have a fallback. If you set `LANG` to `en_US` and the message file

in `/usr/share/locale/en_US/LC_MESSAGES` does not exist, it falls back to `/usr/share/locale/en/LC_MESSAGES`.

A fallback chain can also be defined, for example, for Breton to French or for Galician to Spanish to Portuguese:

```
LANGUAGE="br_FR:fr_FR"
```

```
LANGUAGE="gl_ES:es_ES:pt_PT"
```

If desired, use the Norwegian variants Nynorsk and Bokmål instead (with additional fallback to `no`):

```
LANG="nn_NO"
```

```
LANGUAGE="nn_NO:nb_NO:no"
```

or

```
LANG="nb_NO"
```

```
LANGUAGE="nb_NO:nn_NO:no"
```

Note that in Norwegian, `LC_TIME` is also treated differently.

One problem that can arise is a separator used to delimit groups of digits not being recognized properly. This occurs if `LANG` is set to only a two-letter language code like `de`, but the definition file `glibc` uses is located in `/usr/share/lib/de_DE/LC_NUMERIC`. Thus `LC_NUMERIC` must be set to `de_DE` to make the separator definition visible to the system.

11.4.4 For More Information

- *The GNU C Library Reference Manual*, Chapter « Locales and Internationalization ». It is included in `glibc-info`.
- Markus Kuhn, *UTF-8 and Unicode FAQ for Unix/Linux*, currently at <http://www.cl.cam.ac.uk/~mgk25/unicode.html>.
- *Unicode-Howto* by Bruno Haible, available at <http://tldp.org/HOWTO/Unicode-HOWTO-1.html>.

Dynamic Kernel Device Management with `udev`

The kernel can add or remove almost any device in a running system. Changes in the device state (whether a device is plugged in or removed) need to be propagated to userspace. Devices need to be configured as soon as they are plugged in and recognized. Users of a certain device need to be informed about any changes in this device's recognized state. `udev` provides the needed infrastructure to dynamically maintain the device node files and symbolic links in the `/dev` directory. `udev` rules provide a way to plug external tools into the kernel device event processing. This enables you to customize `udev` device handling by, for example, adding certain scripts to execute as part of kernel device handling, or request and import additional data to evaluate during device handling.

12.1 The `/dev` Directory

The device nodes in the `/dev` directory provide access to the corresponding kernel devices. With `udev`, the `/dev` directory reflects the current state of the kernel. Every kernel device has one corresponding device file. If a device is disconnected from the system, the device node is removed.

The content of the `/dev` directory is kept on a temporary file system and all files are rendered at every system start-up. Manually created or modified files do not, by design, survive a reboot. Static files and directories that should always be present in the `/dev` directory regardless of the state of the corresponding kernel device can be placed in the `/lib/udev/devices` directory. At system start-up, the contents of that directory is copied to the `/dev` directory with the same ownership and permissions as the files in `/lib/udev/devices`.

12.2 Kernel uevents and udev

The required device information is exported by the `sysfs` file system. For every device the kernel has detected and initialized, a directory with the device name is created. It contains attribute files with device-specific properties.

Every time a device is added or removed, the kernel sends a uevent to notify `udev` of the change. The `udev` daemon reads and parses all provided rules from the `/etc/udev/rules.d/*.rules` files once at start-up and keeps them in memory. If rules files are changed, added or removed, the daemon can reload the in-memory representation of all rules with the command `udevadm control reload_rules`. This is also done when running `/etc/init.d/boot.udev reload`. For more details on `udev` rules and their syntax, refer to Section 12.6, « Influencing Kernel Device Event Handling with `udev` Rules » (p. 207).

Every received event is matched against the set of provided rules. The rules can add or change event environment keys, request a specific name for the device node to create, add symlinks pointing to the node or add programs to run after the device node is created. The driver core `uevents` are received from a kernel netlink socket.

12.3 Drivers, Kernel Modules and Devices

The kernel bus drivers probe for devices. For every detected device, the kernel creates an internal device structure while the driver core sends a uevent to the `udev` daemon. Bus devices identify themselves by a specially-formatted ID, which tells what kind of device it is. Usually these IDs consist of vendor and product ID and other subsystem-specific values. Every bus has its own scheme for these IDs, called `MODALIAS`. The kernel takes the device information, composes a `MODALIAS` ID string from it and sends that string along with the event. For a USB mouse, it looks like this:

```
MODALIAS=usb:v046DpC03Ed2000dc00dsc00dp00ic03isc01ip02
```

Every device driver carries a list of known aliases for devices it can handle. The list is contained in the kernel module file itself. The program `depmod` reads the ID lists and creates the file `modules.alias` in the kernel's `/lib/modules` directory

for all currently available modules. With this infrastructure, module loading is as easy as calling `modprobe` for every event that carries a `MODALIAS` key. If `modprobe $MODALIAS` is called, it matches the device alias composed for the device with the aliases provided by the modules. If a matching entry is found, that module is loaded. All this is automatically triggered by `udev`.

12.4 Booting and Initial Device Setup

All device events happening during the boot process before the `udev` daemon is running are lost, because the infrastructure to handle these events resides on the root file system and is not available at that time. To cover that loss, the kernel provides a `uevent` file located in the device directory of every device in the `sysfs` file system. By writing `add` to that file, the kernel resends the same event as the one lost during boot. A simple loop over all `uevent` files in `/sys` triggers all events again to create the device nodes and perform device setup.

As an example, a USB mouse present during boot may not be initialized by the early boot logic, because the driver is not available at that time. The event for the device discovery was lost and failed to find a kernel module for the device. Instead of manually searching for possibly connected devices, `udev` just requests all device events from the kernel after the root file system is available, so the event for the USB mouse device just runs again. Now it finds the kernel module on the mounted root file system and the USB mouse can be initialized.

From userspace, there is no visible difference between a device coldplug sequence and a device discovery during runtime. In both cases, the same rules are used to match and the same configured programs are run.

12.5 Monitoring the Running `udev` Daemon

The program `udevadm monitor` can be used to visualize the driver core events and the timing of the `udev` event processes.

```

UEVENT[1185238505.276660] add    /devices/pci0000:00/0000:00:1d.2/usb3/3-1
(usb)
UDEV  [1185238505.279198] add    /devices/pci0000:00/0000:00:1d.2/usb3/3-1
(usb)
UEVENT[1185238505.279527] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0 (usb)
UDEV  [1185238505.285573] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0 (usb)
UEVENT[1185238505.298878] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0/input/input10 (input)
UDEV  [1185238505.305026] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0/input/input10 (input)
UEVENT[1185238505.305442] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0/input/input10/mouse2 (input)
UEVENT[1185238505.306440] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0/input/input10/event4 (input)
UDEV  [1185238505.325384] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0/input/input10/event4 (input)
UDEV  [1185238505.342257] add    /devices/pci0000:00/0000:00:1d.2/
usb3/3-1/3-1:1.0/input/input10/mouse2 (input)

```

The UEVENT lines show the events the kernel has sent over netlink. The UDEV lines show the finished udev event handlers. The timing is printed in microseconds. The time between UEVENT and UDEV is the time udev took to process this event or the udev daemon has delayed its execution to synchronize this event with related and already running events. For example, events for hard disk partitions always wait for the main disk device event to finish, because the partition events may rely on the data that the main disk event has queried from the hardware.

`udevadm monitor --env` shows the complete event environment:

```

ACTION=add
DEVPATH=/devices/pci0000:00/0000:00:1d.2/usb3/3-1/3-1:1.0/input/input10
SUBSYSTEM=input
SEQNUM=1181
NAME="Logitech USB-PS/2 Optical Mouse"
PHYS="usb-0000:00:1d.2-1/input0"
UNIQ=""
EV=7
KEY=70000 0 0 0 0
REL=103
MODALIAS=input:b0003v046DpC03Ee01110-e0,1,2,k110,111,112,r0,1,8,amlsfw

```

udev also sends messages to syslog. The default syslog priority that controls which messages are sent to syslog is specified in the udev configuration file `/etc/udev/udev.conf`. The log priority of the running daemon can be changed with `udevadm control log_priority=level/number`.

12.6 Influencing Kernel Device Event Handling with udev Rules

A `udev` rule can match any property the kernel adds to the event itself or any information that the kernel exports to `sysfs`. The rule can also request additional information from external programs. Every event is matched against all provided rules. All rules are located in the `/etc/udev/rules.d` directory.

Every line in the rules file contains at least one key value pair. There are two kinds of keys, match and assignment keys. If all match keys match their values, the rule is applied and the assignment keys are assigned the specified value. A matching rule may specify the name of the device node, add symlinks pointing to the node or run a specified program as part of the event handling. If no matching rule is found, the default device node name is used to create the device node. Detailed information about the rule syntax and the provided keys to match or import data are described in the `udev` man page. The following example rules provide a basic introduction to `udev` rule syntax. The example rules are all taken from the `udev` default rule set that is located under `/etc/udev/rules.d/50-udev-default.rules`.

Example 12.1 *Example udev Rules*

```
# console
KERNEL=="console", MODE="0600", OPTIONS="last_rule"

# serial devices
KERNEL=="ttyUSB*", ATTRS{product}=="[Pp]alm*Handheld*", SYMLINK+="pilot"

# printer
SUBSYSTEM=="usb", KERNEL=="lp*", NAME="usb/%k", SYMLINK+="usb%k", GROUP="lp"

# kernel firmware loader
SUBSYSTEM=="firmware", ACTION=="add", RUN+="firmware.sh"
```

The `console` rule consists of three keys: one match key (`KERNEL`) and two assignment keys (`MODE`, `OPTIONS`). The `KERNEL` match rule searches the device list for any items of the type `console`. Only exact matches are valid and trigger this rule to be executed. The `MODE` key assigns special permissions to the device node, in this case, read and write permissions to the owner of this device only. The `OPTIONS` key makes this rule the last rule to be applied to any device of this type. Any later rule matching this particular device type does not have any effect.

The `serial devices` rule is not available in `50-udev-default.rules` anymore, but it is still worth considering. It consists of two match keys (`KERNEL`

and `ATTRS`) and one assign key (`SYMLINK`). The `KERNEL` key searches for all devices of the `ttyUSB` type. Using the `*` wild card, this key matches several of these devices. The second match key, `ATTRS`, checks whether the `product` attribute file in `sysfs` for any `ttyUSB` device contains a certain string. The assign key (`SYMLINK`) triggers the addition of a symbolic link to this device under `/dev/pilot`. The operator used in this key (`+=`) tells `udev` to additionally perform this action, even if previous or later rules add other symbolic links. As this rule contains two match keys, it is only applied if both conditions are met.

The `printer` rule deals with USB printers and contains two match keys which must both apply to get the entire rule applied (`SUBSYSTEM` and `KERNEL`). Three assign keys deal with the naming for this device type (`NAME`), the creation of symbolic device links (`SYMLINK`) and the group membership for this device type (`GROUP`). Using the `*` wild card in the `KERNEL` key makes it match several `lp` printer devices. Substitutions are used in both, the `NAME` and the `SYMLINK` keys to extend these strings by the internal device name. For example, the symlink to the first `lp` USB printer would read `/dev/usblp0`.

The `kernel firmware loader` rule makes `udev` load additional firmware by an external helper script during runtime. The `SUBSYSTEM` match key searches for the `firmware` subsystem. The `ACTION` key checks whether any device belonging to the `firmware` subsystem has been added. The `RUN+=` key triggers the execution of the `firmware.sh` script to locate the firmware that is to be loaded.

Some general characteristics are common to all rules:

- Each rule consists of one or more key value pairs separated by a comma.
- A key's operation is determined by the operator. `udev` rules support several different operators.
- Each given value must be enclosed by quotation marks.
- Each line of the rules file represents one rule. If a rule is longer than just one line, use `\` to join the different lines just as you would do in shell syntax.
- `udev` rules support a shell-style pattern that matches the `*`, `?`, and `[]` patterns.
- `udev` rules support substitutions.

12.6.1 Using Operators in `udev` Rules

Creating keys you can choose from several different operators, depending on the type of key you want to create. Match keys will normally just be used to find a value that either matches or explicitly mismatches the search value. Match keys contain either of the following operators:

`==`

Compare for equality. If the key contains a search pattern, all results matching this pattern are valid.

`!=`

Compare for non-equality. If the key contains a search pattern, all results matching this pattern are valid.

Any of the following operators can be used with assign keys:

`=`

Assign a value to a key. If the key previously consisted of a list of values, the key resets and only the single value is assigned.

`+=`

Add a value to a key that contains a list of entries.

`:=`

Assign a final value. Disallow any later change by later rules.

12.6.2 Using Substitutions in `udev` Rules

`udev` rules support the use of placeholders and substitutions. Use them in a similar fashion as you would do in any other scripts. The following substitutions can be used with `udev` rules:

`%r, $root`

The device directory, `/dev` by default.

`%p, $devpath`

The value of `DEVPATH`.

`%k, $kernel`

The value of `KERNEL` or the internal device name.

`%n, $number`

The device number.

`%N, $tempnode`

The temporary name of the device file.

`%M, $major`

The major number of the device.

`%m, $minor`

The minor number of the device.

`%s{attribute}, $attr{attribute}`

The value of a `sysfs` attribute (specified by *attribute*).

`%E{variable}, $attr{variable}`

The value of an environment variable (specified by *variable*).

`%c, $result`

The output of PROGRAM.

`%%`

The `%` character.

`$$`

The `$` character.

12.6.3 Using `udev` Match Keys

Match keys describe conditions that must be met before a `udev` rule can be applied. The following match keys are available:

`ACTION`

The name of the event action, for example, `add` or `remove` when adding or removing a device.

`DEVPATH`

The device path of the event device, for example, `DEVPATH=/bus/pci/drivers/ipw3945` to search for all events related to the `ipw3945` driver.

`KERNEL`

The internal (kernel) name of the event device.

SUBSYSTEM

The subsystem of the event device, for example, `SUBSYSTEM=usb` for all events related to USB devices.

ATTR{*filename*}

`sysfs` attributes of the event device. To match a string contained in the `vendor` attribute file name, you could use `ATTR{vendor}=="On[sS]tream"`, for example.

KERNELS

Let `udev` search the device path upwards for a matching device name.

SUBSYSTEMS

Let `udev` search the device path upwards for a matching device subsystem name.

DRIVERS

Let `udev` search the device path upwards for a matching device driver name.

ATTRS{*filename*}

Let `udev` search the device path upwards for a device with matching `sysfs` attribute values.

ENV{*key*}

The value of an environment variable, for example, `ENV{ID_BUS}="ieee1394` to search for all events related to the FireWire bus ID.

PROGRAM

Let `udev` execute an external program. To be successful, the program must return with exit code zero. The program's output, printed to `stdout`, is available to the `RESULT` key.

RESULT

Match the output string of the last `PROGRAM` call. Either include this key in the same rule as the `PROGRAM` key or in a later one.

12.6.4 Using `udev` Assign Keys

In contrast to the match keys described above, assign keys do not describe conditions that must be met. They assign values, names and actions to the device nodes maintained by `udev`.

NAME

The name of the device node to be created. Once a rule has set a node name, all other rules with a `NAME` key for this node are ignored.

SYMLINK

The name of a symlink related to the node to be created. Multiple matching rules can add symlinks to be created with the device node. You can also specify multiple symlinks for one node in one rule using the space character to separate the symlink names.

OWNER, GROUP, MODE

The permissions for the new device node. Values specified here overwrite anything that has been compiled in.

ATTR{key}

Specify a value to be written to a `sysfs` attribute of the event device. If the `==` operator is used, this key is also used to match against the value of a `sysfs` attribute.

ENV{key}

Tell `udev` to export a variable to the environment. If the `==` operator is used, this key is also used to match against an environment variable.

RUN

Tell `udev` to add a program to the list of programs to be executed for this device. Keep in mind to restrict this to very short tasks to avoid blocking further events for this device.

LABEL

Add a label where a `GOTO` can jump to.

GOTO

Tell `udev` to skip a number of rules and continue with the one that carries the label referenced by the `GOTO` key.

IMPORT{type}

Load variables into the event environment such as the output of an external program. `udev` imports variables of several different types. If no type is specified, `udev` tries to determine the type itself based on the executable bit of the file permissions.

- `program` tells `udev` to execute an external program and import its output.

- `file` tells `udev` to import a text file.
- `parent` tells `udev` to import the stored keys from the parent device.

WAIT_FOR_SYSFS

Tells `udev` to wait for the specified `sysfs` file to be created for a certain device. For example, `WAIT_FOR_SYSFS="ioerr_cnt"` informs `udev` to wait until the `ioerr_cnt` file has been created.

OPTIONS

The `OPTION` key may have several possible values:

- `last_rule` tells `udev` to ignore all later rules.
- `ignore_device` tells `udev` to ignore this event completely.
- `ignore_remove` tells `udev` to ignore all later remove events for the device.
- `all_partitions` tells `udev` to create device nodes for all available partitions on a block device.

12.7 Persistent Device Naming

The dynamic device directory and the `udev` rules infrastructure make it possible to provide stable names for all disk devices—regardless of their order of recognition or the connection used for the device. Every appropriate block device the kernel creates is examined by tools with special knowledge about certain buses, drive types or file systems. Along with the dynamic kernel-provided device node name, `udev` maintains classes of persistent symbolic links pointing to the device:

```
/dev/disk
|-- by-id
|   |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B -> ../../sda
|   |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B-part1 -> ../../sda1
|   |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B-part6 -> ../../sda6
|   |-- scsi-SATA_HTS726060M9AT00_MRH453M4HWHG7B-part7 -> ../../sda7
|   |-- usb-Generic_STORAGE_DEVICE_02773 -> ../../sdd
|   `-- usb-Generic_STORAGE_DEVICE_02773-part1 -> ../../sdd1
|-- by-label
|   |-- Photos -> ../../sdd1
|   |-- SUSE10 -> ../../sda7
|   `-- devel -> ../../sda6
|-- by-path
```

```

| |-- pci-0000:00:1f.2-scsi-0:0:0:0 -> ../../sda
| |-- pci-0000:00:1f.2-scsi-0:0:0:0-part1 -> ../../sda1
| |-- pci-0000:00:1f.2-scsi-0:0:0:0-part6 -> ../../sda6
| |-- pci-0000:00:1f.2-scsi-0:0:0:0-part7 -> ../../sda7
| |-- pci-0000:00:1f.2-scsi-1:0:0:0 -> ../../sr0
| |-- usb-02773:0:0:2 -> ../../sdd
| |-- usb-02773:0:0:2-part1 -> ../../sdd1
|-- by-uuid
| |-- 159a47a4-e6e6-40be-a757-a629991479ae -> ../../sda7
| |-- 3e999973-00c9-4917-9442-b7633bd95b9e -> ../../sda6
`-- 4210-8F8C -> ../../sdd1

```

12.8 Files used by udev

`/sys/*`

Virtual file system provided by the Linux kernel, exporting all currently known devices. This information is used by `udev` to create device nodes in `/dev`

`/dev/*`

Dynamically created device nodes and static content copied at boot time from `/lib/udev/devices/*`

The following files and directories contain the crucial elements of the `udev` infrastructure:

`/etc/udev/udev.conf`

Main `udev` configuration file.

`/etc/udev/rules.d/*`

`udev` event matching rules.

`/lib/udev/devices/*`

Static `/dev` content.

`/lib/udev/*`

Helper programs called from `udev` rules.

12.9 For More Information

For more information about the `udev` infrastructure, refer to the following man pages:

udev

General information about udev, keys, rules and other important configuration issues.

udevadm

udevadm can be used to control the runtime behavior of udev, request kernel events, manage the event queue and provide simple debugging mechanisms.

udevd

Information about the udev event managing daemon.

Partie IV. Services

Basic Networking

Linux offers the necessary networking tools and features for integration into all types of network structures. Network access using a network card, modem or other device can be configured with YaST. Manual configuration is also possible. In this chapter only the fundamental mechanisms and the relevant network configuration files are covered.

Linux and other Unix operating systems use the TCP/IP protocol. It is not a single network protocol, but a family of network protocols that offer various services. The protocols listed in Tableau 13.1, « Several Protocols in the TCP/IP Protocol Family » (p. 219), are provided for the purpose of exchanging data between two machines via TCP/IP. Networks combined by TCP/IP, comprising a worldwide network, are also referred to as « the Internet. »

RFC stands for *Request for Comments*. RFCs are documents that describe various Internet protocols and implementation procedures for the operating system and its applications. The RFC documents describe the setup of Internet protocols. To expand your knowledge of any of the protocols, refer to the appropriate RFC documents. These are available at <http://www.ietf.org/rfc.html>.

Tableau 13.1 *Several Protocols in the TCP/IP Protocol Family*

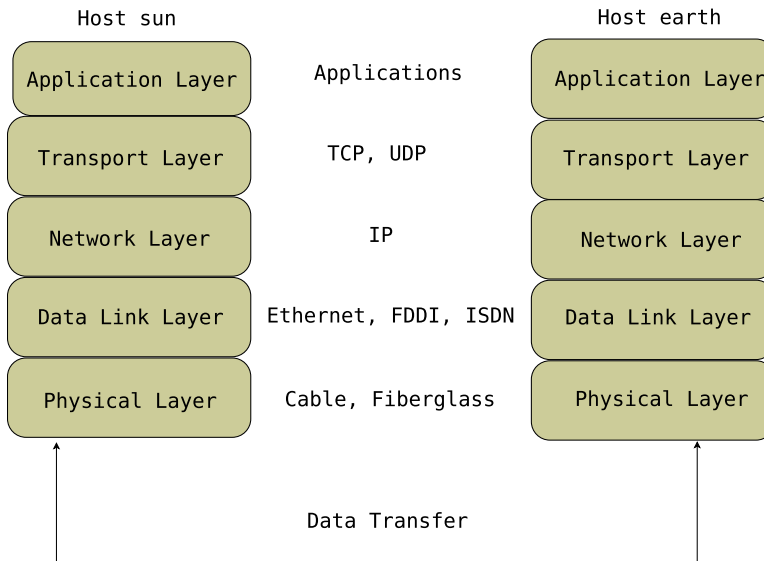
Protocol	Description
TCP	Transmission Control Protocol: a connection-oriented secure protocol. The data to transmit is first sent by the application as a stream of data and

Protocol	Description
	<p>converted into the appropriate format by the operating system. The data arrives at the respective application on the destination host in the original data stream format it was initially sent. TCP determines whether any data has been lost or jumbled during the transmission. TCP is implemented wherever the data sequence matters.</p>
UDP	<p>User Datagram Protocol: a connectionless, insecure protocol. The data to transmit is sent in the form of packets generated by the application. The order in which the data arrives at the recipient is not guaranteed and data loss is possible. UDP is suitable for record-oriented applications. It features a smaller latency period than TCP.</p>
ICMP	<p>Internet Control Message Protocol: Essentially, this is not a protocol for the end user, but a special control protocol that issues error reports and can control the behavior of machines participating in TCP/IP data transfer. In addition, it provides a special echo mode that can be viewed using the program ping.</p>
IGMP	<p>Internet Group Management Protocol: This protocol controls machine behavior when implementing IP multicast.</p>

As shown in Figure 13.1, « Simplified Layer Model for TCP/IP » (p. 221), data exchange takes place in different layers. The actual network layer is the insecure data

transfer via IP (Internet protocol). On top of IP, TCP (transmission control protocol) guarantees, to a certain extent, security of the data transfer. The IP layer is supported by the underlying hardware-dependent protocol, such as ethernet.

Figure 13.1 *Simplified Layer Model for TCP/IP*



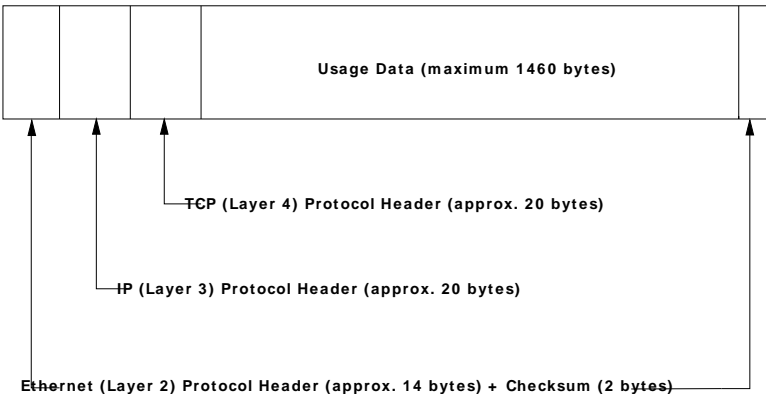
The diagram provides one or two examples for each layer. The layers are ordered according to *abstraction levels*. The lowest layer is very close to the hardware. The uppermost layer, however, is almost a complete abstraction from the hardware. Every layer has its own special function. The special functions of each layer are mostly implicit in their description. The data link and physical layers represent the physical network used, such as ethernet.

Almost all hardware protocols work on a packet-oriented basis. The data to transmit is collected into *packets* (it cannot be sent all at once). The maximum size of a TCP/IP packet is approximately 64 KB. Packets are normally quite smaller, as the network hardware can be a limiting factor. The maximum size of a data packet on an ethernet is about fifteen hundred bytes. The size of a TCP/IP packet is limited to this amount when the data is sent over an ethernet. If more data is transferred, more data packets need to be sent by the operating system.

For the layers to serve their designated functions, additional information regarding each layer must be saved in the data packet. This takes place in the *header* of the

packet. Every layer attaches a small block of data, called the protocol header, to the front of each emerging packet. A sample TCP/IP data packet traveling over an ethernet cable is illustrated in Figure 13.2, « TCP/IP Ethernet Packet » (p. 222). The proof sum is located at the end of the packet, not at the beginning. This simplifies things for the network hardware.

Figure 13.2 *TCP/IP Ethernet Packet*



When an application sends data over the network, the data passes through each layer, all implemented in the Linux kernel except the physical layer. Each layer is responsible for preparing the data so it can be passed to the next layer. The lowest layer is ultimately responsible for sending the data. The entire procedure is reversed when data is received. Like the layers of an onion, in each layer the protocol headers are removed from the transported data. Finally, the transport layer is responsible for making the data available for use by the applications at the destination. In this manner, one layer only communicates with the layer directly above or below it. For applications, it is irrelevant whether data is transmitted via a 100 Mbit/s FDDI network or via a 56-Kbit/s modem line. Likewise, it is irrelevant for the data line which kind of data is transmitted, as long as packets are in the correct format.

13.1 IP Addresses and Routing

The discussion in this section is limited to IPv4 networks. For information about IPv6 protocol, the successor to IPv4, refer to Section 13.2, « IPv6—The Next Generation Internet » (p. 225).

13.1.1 IP Addresses

Every computer on the Internet has a unique 32-bit address. These 32 bits (or 4 bytes) are normally written as illustrated in the second row in Exemple 13.1, « Writing IP Addresses » (p. 223).

Example 13.1 *Writing IP Addresses*

```
IP Address (binary): 11000000 10101000 00000000 00010100
IP Address (decimal): 192. 168. 0. 20
```

In decimal form, the four bytes are written in the decimal number system, separated by periods. The IP address is assigned to a host or a network interface. It can be used only once throughout the world. There are exceptions to this rule, but these are not relevant to the following passages.

The points in IP addresses indicate the hierarchical system. Until the 1990s, IP addresses were strictly categorized in classes. However, this system proved too inflexible and was discontinued. Now, *classless routing* (CIDR, classless interdomain routing) is used.

13.1.2 Netmasks and Routing

Netmasks are used to define the address range of a subnetwork. If two hosts are in the same subnetwork, they can reach each other directly. If they are not in the same subnetwork, they need the address of a gateway that handles all the traffic for the subnetwork. To check if two IP addresses are in the same subnet, simply « AND » both addresses with the netmask. If the result is identical, both IP addresses are in the same local network. If there are differences, the remote IP address, and thus the remote interface, can only be reached over a gateway.

To understand how the netmask works, look at Exemple 13.2, « Linking IP Addresses to the Netmask » (p. 224). The netmask consists of 32 bits that identify how much of an IP address belongs to the network. All those bits that are 1 mark the corresponding bit in the IP address as belonging to the network. All bits that are 0 mark bits inside the subnetwork. This means that the more bits are 1, the smaller the subnetwork is. Because the netmask always consists of several successive 1 bits, it is also possible to just count the number of bits in the netmask. In Exemple 13.2, « Linking IP Addresses to the Netmask » (p. 224) the first net with 24 bits could also be written as 192.168.0.0/24.

Example 13.2 *Linking IP Addresses to the Netmask*

```
IP address (192.168.0.20): 11000000 10101000 00000000 00010100
Netmask (255.255.255.0): 11111111 11111111 11111111 00000000
-----
Result of the link:      11000000 10101000 00000000 00000000
In the decimal system:   192.      168.      0.      0

IP address (213.95.15.200): 11010101 10111111 00001111 11001000
Netmask (255.255.255.0): 11111111 11111111 11111111 00000000
-----
Result of the link:      11010101 10111111 00001111 00000000
In the decimal system:   213.      95.      15.      0
```

To give another example: all machines connected with the same ethernet cable are usually located in the same subnetwork and are directly accessible. Even when the subnet is physically divided by switches or bridges, these hosts can still be reached directly.

IP addresses outside the local subnet can only be reached if a gateway is configured for the target network. In the most common case, there is only one gateway that handles all traffic that is external. However, it is also possible to configure several gateways for different subnets.

If a gateway has been configured, all external IP packets are sent to the appropriate gateway. This gateway then attempts to forward the packets in the same manner—from host to host—until it reaches the destination host or the packet's TTL (time to live) expires.

Tableau 13.2 *Specific Addresses*

Address Type	Description
Base Network Address	This is the netmask AND any address in the network, as shown in Exemple 13.2, « Linking IP Addresses to the Netmask » (p. 224) under Result. This address cannot be assigned to any hosts.
Broadcast Address	This basically says, « Access all hosts in this subnetwork. » To gene-

Address Type	Description
	rate this, the netmask is inverted in binary form and linked to the base network address with a logical OR. The above example therefore results in 192.168.0.255. This address cannot be assigned to any hosts.
Local Host	The address 127.0.0.1 is assigned to the « loopback device » on each host. A connection can be set up to your own machine with this address and with all addresses from the complete 127.0.0.0/8 loopback network as defined with IPv4. With IPv6 there is just one loopback address (:::1).

Because IP addresses must be unique all over the world, you cannot just select random addresses. There are three address domains to use if you want to set up a private IP-based network. These cannot get any connection from the rest of the Internet, because they cannot be transmitted over the Internet. These address domains are specified in RFC 1597 and listed in Tableau 13.3, « Private IP Address Domains » (p. 225).

Tableau 13.3 *Private IP Address Domains*

Network/Netmask	Domain
10.0.0.0/255.0.0.0	10.x.x.x
172.16.0.0/255.240.0.0	172.16.x.x – 172.31.x.x
192.168.0.0/255.255.0.0	192.168.x.x

13.2 IPv6—The Next Generation Internet

Due to the emergence of the WWW (World Wide Web), the Internet has experienced explosive growth, with an increasing number of computers communicating via TCP/IP in the past fifteen years. Since Tim Berners-Lee at CERN (<http://public.web.cern.ch>) invented the WWW in 1990, the number of Internet hosts has grown from a few thousand to about a hundred million.

As mentioned, an IPv4 address consists of only 32 bits. Also, quite a few IP addresses are lost—they cannot be used due to the way in which networks are organized. The number of addresses available in your subnet is two to the power of the number of bits, minus two. A subnetwork has, for example, 2, 6, or 14 addresses available. To connect 128 hosts to the Internet, for example, you need a subnetwork with 256 IP addresses, from which only 254 are usable, because two IP addresses are needed for the structure of the subnetwork itself: the broadcast and the base network address.

Under the current IPv4 protocol, DHCP or NAT (network address translation) are the typical mechanisms used to circumvent the potential address shortage. Combined with the convention to keep private and public address spaces separate, these methods can certainly mitigate the shortage. The problem with them lies in their configuration, which is a chore to set up and a burden to maintain. To set up a host in an IPv4 network, you need a number of address items, such as the host's own IP address, the subnetmask, the gateway address and maybe a name server address. All these items need to be known and cannot be derived from somewhere else.

With IPv6, both the address shortage and the complicated configuration should be a thing of the past. The following sections tell more about the improvements and benefits brought by IPv6 and about the transition from the old protocol to the new one.

13.2.1 Advantages

The most important and most visible improvement brought by the new protocol is the enormous expansion of the available address space. An IPv6 address is made up of 128 bit values instead of the traditional 32 bits. This provides for as many as several quadrillion IP addresses.

However, IPv6 addresses are not only different from their predecessors with regard to their length. They also have a different internal structure that may contain more specific information about the systems and the networks to which they belong. More details about this are found in Section 13.2.2, « Address Types and Structure » (p. 228).

The following is a list of some other advantages of the new protocol:

Autoconfiguration

IPv6 makes the network « plug and play » capable, which means that a newly set up system integrates into the (local) network without any manual configuration. The new host uses its automatic configuration mechanism to derive its own address from the information made available by the neighboring routers, relying on a protocol called the *neighbor discovery* (ND) protocol. This method does not require any intervention on the administrator's part and there is no need to maintain a central server for address allocation—an additional advantage over IPv4, where automatic address allocation requires a DHCP server or the usage of ARP and 169.254.0.0/16 addresses.

Nevertheless if a router is connected to a switch, the router should send periodic advertisements with flags telling the hosts of a network how they should interact with each other. For more information, see RFC 2462 and the `radvd.conf(5)` manpage, and RFC 3315.

Mobility

IPv6 makes it possible to assign several addresses to one network interface at the same time. This allows users to access several networks easily, something that could be compared with the international roaming services offered by mobile phone companies: when you take your mobile phone abroad, the phone automatically logs in to a foreign service as soon as it enters the corresponding area, so you can be reached under the same number everywhere and are able to place an outgoing call just like in your home area.

Secure Communication

With IPv4, network security is an add-on function. IPv6 includes IPsec as one of its core features, allowing systems to communicate over a secure tunnel to avoid eavesdropping by outsiders on the Internet.

Backward Compatibility

Realistically, it would be impossible to switch the entire Internet from IPv4 to IPv6 at one time. Therefore, it is crucial that both protocols are able to coexist not only on the Internet, but also on one system. This is ensured by compatible addresses (IPv4 addresses can easily be translated into IPv6 addresses) and through the use of a number of tunnels. See Section 13.2.3, « Coexistence of IPv4 and IPv6 » (p. 232). Also, systems can rely on a *dual stack IP* technique to support both protocols at the same time, meaning that they have two network stacks that are completely separate, such that there is no interference between the two protocol versions.

Custom Tailored Services through Multicasting

With IPv4, some services, such as SMB, need to broadcast their packets to all hosts in the local network. IPv6 allows a much more fine-grained approach by enabling servers to address hosts through *multicasting*—by addressing a number of hosts as parts of a group (which is different from addressing all hosts through *broadcasting* or each host individually through *unicasting*). Which hosts are addressed as a group may depend on the concrete application. There are some predefined groups to address all name servers (the *all name servers multicast group*), for example, or all routers (the *all routers multicast group*).

13.2.2 Address Types and Structure

As mentioned, the current IP protocol is lacking in two important aspects: there is an increasing shortage of IP addresses and configuring the network and maintaining the routing tables is becoming a more complex and burdensome task. IPv6 solves the first problem by expanding the address space to 128 bits. The second one is countered by introducing a hierarchical address structure, combined with sophisticated techniques to allocate network addresses, as well as *multihoming* (the ability to assign several addresses to one device, giving access to several networks).

When dealing with IPv6, it is useful to know about three different types of addresses:

Unicast

Addresses of this type are associated with exactly one network interface. Packets with such an address are delivered to only one destination. Accordingly, unicast addresses are used to transfer packets to individual hosts on the local network or the Internet.

Multicast

Addresses of this type relate to a group of network interfaces. Packets with such an address are delivered to all destinations that belong to the group. Multicast addresses are mainly used by certain network services to communicate with certain groups of hosts in a well-directed manner.

Anycast

Addresses of this type are related to a group of interfaces. Packets with such an address are delivered to the member of the group that is closest to the sender, according to the principles of the underlying routing protocol. Anycast addresses are used to make it easier for hosts to find out about servers offering certain services in the given network area. All servers of the same type have the same

anycast address. Whenever a host requests a service, it receives a reply from the server with the closest location, as determined by the routing protocol. If this server should fail for some reason, the protocol automatically selects the second closest server, then the third one, and so forth.

An IPv6 address is made up of eight four-digit fields, each representing 16 bits, written in hexadecimal notation. They are separated by colons (:). Any leading zero bytes within a given field may be dropped, but zeros within the field or at its end may not. Another convention is that more than four consecutive zero bytes may be collapsed into a double colon. However, only one such :: is allowed per address. This kind of shorthand notation is shown in Example 13.3, « Sample IPv6 Address » (p. 229), where all three lines represent the same address.

Example 13.3 *Sample IPv6 Address*

```
fe80 : 0000 : 0000 : 0000 : 0000 : 10 : 1000 : 1a4
fe80 :    0 :    0 :    0 :    0 : 10 : 1000 : 1a4
fe80 :                                : 10 : 1000 : 1a4
```

Each part of an IPv6 address has a defined function. The first bytes form the prefix and specify the type of address. The center part is the network portion of the address, but it may be unused. The end of the address forms the host part. With IPv6, the netmask is defined by indicating the length of the prefix after a slash at the end of the address. An address, as shown in Example 13.4, « IPv6 Address Specifying the Prefix Length » (p. 229), contains the information that the first 64 bits form the network part of the address and the last 64 form its host part. In other words, the 64 means that the netmask is filled with 64 1-bit values from the left. Just like with IPv4, the IP address is combined with AND with the values from the netmask to determine whether the host is located in the same subnetwork or in another one.

Example 13.4 *IPv6 Address Specifying the Prefix Length*

```
fe80::10:1000:1a4/64
```

IPv6 knows about several predefined types of prefixes. Some of these are shown in Tableau 13.4, « Various IPv6 Prefixes » (p. 229).

Tableau 13.4 *Various IPv6 Prefixes*

Prefix (hex)	Definition
00	IPv4 addresses and IPv4 over IPv6 compatibility addresses. These

Prefix (hex)	Definition
	are used to maintain compatibility with IPv4. Their use still requires a router able to translate IPv6 packets into IPv4 packets. Several special addresses, such as the one for the loopback device, have this prefix as well.
2 or 3 as the first digit	Aggregatable global unicast addresses. As is the case with IPv4, an interface can be assigned to form part of a certain subnetwork. Currently, there are the following address spaces: <code>2001::/16</code> (production quality address space) and <code>2002::/16</code> (6to4 address space).
<code>fe80::/10</code>	Link-local addresses. Addresses with this prefix should not be routed and should therefore only be reachable from within the same subnetwork.
<code>fec0::/10</code>	Site-local addresses. These may be routed, but only within the network of the organization to which they belong. In effect, they are the IPv6 equivalent of the current private network address space, such as <code>10.x.x.x</code> .
<code>ff</code>	These are multicast addresses.

A unicast address consists of three basic components:

Public Topology

The first part (which also contains one of the prefixes mentioned above) is used to route packets through the public Internet. It includes information about the company or institution that provides the Internet access.

Site Topology

The second part contains routing information about the subnetwork to which to deliver the packet.

Interface ID

The third part identifies the interface to which to deliver the packet. This also allows for the MAC to form part of the address. Given that the MAC is a globally unique, fixed identifier coded into the device by the hardware maker, the configuration procedure is substantially simplified. In fact, the first 64 address bits are consolidated to form the `EUI-64` token, with the last 48 bits taken from the MAC, and the remaining 24 bits containing special information about the token type. This also makes it possible to assign an `EUI-64` token to interfaces that do not have a MAC, such as those based on PPP or ISDN.

On top of this basic structure, IPv6 distinguishes between five different types of unicast addresses:

`::` (unspecified)

This address is used by the host as its source address when the interface is initialized for the first time—when the address cannot yet be determined by other means.

`::1` (loopback)

The address of the loopback device.

IPv4 Compatible Addresses

The IPv6 address is formed by the IPv4 address and a prefix consisting of 96 zero bits. This type of compatibility address is used for tunneling (see Section 13.2.3, « Coexistence of IPv4 and IPv6 » (p. 232)) to allow IPv4 and IPv6 hosts to communicate with others operating in a pure IPv4 environment.

IPv4 Addresses Mapped to IPv6

This type of address specifies a pure IPv4 address in IPv6 notation.

Local Addresses

There are two address types for local use:

link-local

This type of address can only be used in the local subnetwork. Packets with a source or target address of this type should not be routed to the Internet or other subnetworks. These addresses contain a special prefix (`fe80::/10`) and the interface ID of the network card, with the middle part consisting of

zero bytes. Addresses of this type are used during automatic configuration to communicate with other hosts belonging to the same subnetwork.

site-local

Packets with this type of address may be routed to other subnetworks, but not to the wider Internet—they must remain inside the organization's own network. Such addresses are used for intranets and are an equivalent of the private address space defined by IPv4. They contain a special prefix ($\text{fec0}::/10$), the interface ID, and a 16 bit field specifying the subnetwork ID. Again, the rest is filled with zero bytes.

As a completely new feature introduced with IPv6, each network interface normally gets several IP addresses, with the advantage that several networks can be accessed through the same interface. One of these networks can be configured completely automatically using the MAC and a known prefix with the result that all hosts on the local network can be reached as soon as IPv6 is enabled (using the link-local address). With the MAC forming part of it, any IP address used in the world is unique. The only variable parts of the address are those specifying the *site topology* and the *public topology*, depending on the actual network in which the host is currently operating.

For a host to go back and forth between different networks, it needs at least two addresses. One of them, the *home address*, not only contains the interface ID but also an identifier of the home network to which it normally belongs (and the corresponding prefix). The home address is a static address and, as such, it does not normally change. Still, all packets destined to the mobile host can be delivered to it, regardless of whether it operates in the home network or somewhere outside. This is made possible by the completely new features introduced with IPv6, such as *stateless auto-configuration* and *neighbor discovery*. In addition to its home address, a mobile host gets one or more additional addresses that belong to the foreign networks where it is roaming. These are called *care-of* addresses. The home network has a facility that forwards any packets destined to the host when it is roaming outside. In an IPv6 environment, this task is performed by the *home agent*, which takes all packets destined to the home address and relays them through a tunnel. On the other hand, those packets destined to the care-of address are directly transferred to the mobile host without any special detours.

13.2.3 Coexistence of IPv4 and IPv6

The migration of all hosts connected to the Internet from IPv4 to IPv6 is a gradual process. Both protocols will coexist for some time to come. The coexistence on one

system is guaranteed where there is a *dual stack* implementation of both protocols. That still leaves the question of how an IPv6 enabled host should communicate with an IPv4 host and how IPv6 packets should be transported by the current networks, which are predominantly IPv4 based. The best solutions offer tunneling and compatibility addresses (see Section 13.2.2, « Address Types and Structure » (p. 228)).

IPv6 hosts that are more or less isolated in the (worldwide) IPv4 network can communicate through tunnels: IPv6 packets are encapsulated as IPv4 packets to move them across an IPv4 network. Such a connection between two IPv4 hosts is called a *tunnel*. To achieve this, packets must include the IPv6 destination address (or the corresponding prefix) as well as the IPv4 address of the remote host at the receiving end of the tunnel. A basic tunnel can be configured manually according to an agreement between the hosts' administrators. This is also called *static tunneling*.

However, the configuration and maintenance of static tunnels is often too labor-intensive to use them for daily communication needs. Therefore, IPv6 provides for three different methods of *dynamic tunneling*:

6over4

IPv6 packets are automatically encapsulated as IPv4 packets and sent over an IPv4 network capable of multicasting. IPv6 is tricked into seeing the whole network (Internet) as a huge local area network (LAN). This makes it possible to determine the receiving end of the IPv4 tunnel automatically. However, this method does not scale very well and is also hampered by the fact that IP multicasting is far from widespread on the Internet. Therefore, it only provides a solution for smaller corporate or institutional networks where multicasting can be enabled. The specifications for this method are laid down in RFC 2529.

6to4

With this method, IPv4 addresses are automatically generated from IPv6 addresses, enabling isolated IPv6 hosts to communicate over an IPv4 network. However, a number of problems have been reported regarding the communication between those isolated IPv6 hosts and the Internet. The method is described in RFC 3056.

IPv6 Tunnel Broker

This method relies on special servers that provide dedicated tunnels for IPv6 hosts. It is described in RFC 3053.

13.2.4 Configuring IPv6

To configure IPv6, you normally do not need to make any changes on the individual workstations. IPv6 is enabled by default. You can disable it during installation in the network configuration step described in Section 1.14.2.2, « Configuration du réseau » (p. 32). To disable or enable IPv6 on an installed system, use the YaST *Network Settings* module. On the *Global Options* tab, check or uncheck the *Enable IPv6* option as necessary. To enable or disable IPv6 manually, edit `/etc/modprobe.d/50-ipv6.conf` and restart the system. If you want to enable it temporarily until the next reboot, enter `modprobe -i ipv6` as root. It is basically impossible to unload the `ipv6` module once loaded.

Because of the autoconfiguration concept of IPv6, the network card is assigned an address in the *link-local* network. Normally, no routing table management takes place on a workstation. The network routers can be queried by the workstation, using the *router advertisement protocol*, for what prefix and gateways should be implemented. The `radvd` program can be used to set up an IPv6 router. This program informs the workstations which prefix to use for the IPv6 addresses and which routers. Alternatively, use `zebra/quagga` for automatic configuration of both addresses and routing.

Consult the `ifcfg-tunnel (5)` man page to get information about how to set up various types of tunnels using the `/etc/sysconfig/network` files.

13.2.5 For More Information

The above overview does not cover the topic of IPv6 comprehensively. For a more in-depth look at the new protocol, refer to the following online documentation and books:

<http://www.ipv6.org/>

The starting point for everything about IPv6.

<http://www.ipv6day.org>

All information needed to start your own IPv6 network.

<http://www.ipv6-to-standard.org/>

The list of IPv6-enabled products.

<http://www.bieringer.de/linux/IPv6/>

Here, find the Linux IPv6-HOWTO and many links related to the topic.

RFC 2640

The fundamental RFC about IPv6.

IPv6 Essentials

A book describing all the important aspects of the topic is *IPv6 Essentials* by Silvia Hagen (ISBN 0-596-00125-8).

13.3 Name Resolution

DNS assists in assigning an IP address to one or more names and assigning a name to an IP address. In Linux, this conversion is usually carried out by a special type of software known as *bind*. The machine that takes care of this conversion is called a *name server*. The names make up a hierarchical system in which each name component is separated by a period. The name hierarchy is, however, independent of the IP address hierarchy described above.

Consider a complete name, such as `jupiter.example.com`, written in the format `hostname.domain`. A full name, referred to as a *fully qualified domain name* (FQDN), consists of a hostname and a domain name (`example.com`). The latter also includes the *top level domain* or TLD (`com`).

TLD assignment has become quite confusing for historical reasons. Traditionally, three-letter domain names are used in the USA. In the rest of the world, the two-letter ISO national codes are the standard. In addition to that, longer TLDs were introduced in 2000 that represent certain spheres of activity (for example, `.info`, `.name`, `.museum`).

In the early days of the Internet (before 1990), the file `/etc/hosts` was used to store the names of all the machines represented over the Internet. This quickly proved to be impractical in the face of the rapidly growing number of computers connected to the Internet. For this reason, a decentralized database was developed to store the hostnames in a widely distributed manner. This database, similar to the name server, does not have the data pertaining to all hosts in the Internet readily available, but can dispatch requests to other name servers.

The top of the hierarchy is occupied by *root name servers*. These root name servers manage the top level domains and are run by the Network Information Center (NIC). Each root name server knows about the name servers responsible for a given top

level domain. Information about top level domain NICs is available at <http://www.internic.net>.

DNS can do more than just resolve hostnames. The name server also knows which host is receiving e-mails for an entire domain—the *mail exchanger (MX)*.

For your machine to resolve an IP address, it must know about at least one name server and its IP address. Easily specify such a name server with the help of YaST. If you have a modem dial-up connection, you may not need to configure a name server manually at all. The dial-up protocol provides the name server address as the connection is made. The configuration of name server access with openSUSE® is described in Section 13.4.1.4, « Configuring Hostname and DNS » (p. 245). Setting up your own name server is described in Chapitre 15, *The Domain Name System* (p. 283).

The protocol `whois` is closely related to DNS. With this program, quickly find out who is responsible for any given domain.

REMARQUE : MDNS and .local Domain Names

The `.local` top level domain is treated as link-local domain by the resolver. DNS requests are sent as multicast DNS requests instead of normal DNS requests. If you already use the `.local` domain in your nameserver configuration, you must switch this option off in `/etc/host.conf`. For more information, see the `host.conf` manual page.

If you want to switch off MDNS during installation, use `nomdns=1` as a boot parameter.

For more information on multicast DNS, see <http://www.multicastdns.org>.

13.4 Configuring a Network Connection with YaST

There are many supported networking types on Linux. Most of them use different device names and the configuration files are spread over several locations in the file system. For a detailed overview of the aspects of manual network configuration, see Section 13.6, « Configuring a Network Connection Manually » (p. 260).

During installation on a laptop (where NetworkManager is active by default) YaST configures all interfaces that have been detected. If NetworkManager is not active, only the first interface with link up (with a network cable connected) is automatically configured. Additional hardware can be configured any time on the installed system. The following sections describe the network configuration for all types of network connections supported by openSUSE.

13.4.1 Configuring the Network Card with YaST

To configure your wired or wireless network card in YaST, select *Network Devices > Network Settings*. After starting the module, YaST displays the *Network Settings* dialog with four tabs: *Global Options*, *Overview*, *Hostname/DNS* and *Routing*.

The *Global Options* tab allows you to set general networking options such as the use of NetworkManager, IPv6 and general DHCP options. For more information, see Section 13.4.1.1, « Configuring Global Networking Options » (p. 238).

The *Overview* tab contains information about installed network interfaces and configurations. Any properly detected network card is listed with its name. You can manually configure new cards, remove or change their configuration in this dialog. If you want to manually configure a card that was not automatically detected, see Section 13.4.1.3, « Configuring an Undetected Network Card » (p. 244). If you want to change the configuration of an already configured card, see Section 13.4.1.2, « Changing the Configuration of a Network Card » (p. 239).

The *Hostname/DNS* tab allows to set the hostname of the machine and name the servers to be used. For more information, see Section 13.4.1.4, « Configuring Hostname and DNS » (p. 245).

The *Routing* tab is used for the configuration of routing. See Section 13.4.1.5, « Configuring Routing » (p. 247) for more information.

Figure 13.3 *Configuring Network Settings*



13.4.1.1 Configuring Global Networking Options

The *Global Options* tab of the YaST *Network Settings* module allows you to set important global networking options, such as the use of NetworkManager, IPv6 and DHCP client options. These settings are applicable for all network interfaces.

In the *Network Setup Method* choose the way network connections are managed. If you want a NetworkManager desktop applet to manage connections for all interfaces, choose *User Controlled with NetworkManager*. This option is well suited for switching between multiple wired and wireless networks. If you do not run a desktop environment (GNOME or KDE), or if your computer is a Xen server, virtual system, or provides network services such as DHCP or DNS in your network, use the *Traditional Method with ifup*. If NetworkManager is used, `nm-applet` should be used to configure network options and the *Overview*, *Hostname/DNS* and *Routing* tabs of the *Network Settings* module are disabled. For more information on NetworkManager, see Chapitre 25, *Using NetworkManager* (p. 437).

In the *IPv6 Protocol Settings* choose whether you want to use the IPv6 protocol. It is possible to use IPv6 together with IPv4. By default, IPv6 is activated. However, in

networks not using IPv6 protocol, response times can be faster with IPv6 protocol disabled. If you want to disable IPv6, uncheck the *Enable IPv6* option. This disables autoload of the kernel module for IPv6. This will be applied after reboot.

In the *DHCP Client Options* configure options for the DHCP client. The *DHCP Client Identifier* must be different for each DHCP client on a single network. If left empty, it defaults to the hardware address of the network interface. However, if you are running several virtual machines using the same network interface and, therefore, the same hardware address, specify a unique free-form identifier here.

The *Hostname to Send* specifies a string used for the hostname option field when dhcpcd sends messages to DHCP server. Some DHCP servers update name server zones (forward and reverse records) according to this hostname (Dynamic DNS). Also, some DHCP servers require the *Hostname to Send* option field to contain a specific string in the DHCP messages from clients. Leave `AUTO` to send the current hostname (that is the one defined in `/etc/HOSTNAME`). Leave the option field empty for not sending any hostname. If you do not want to change the default route according to the information from DHCP, uncheck *Change Default Route via DHCP*.

13.4.1.2 Changing the Configuration of a Network Card

To change the configuration of a network card, select a card from the list of the detected cards in *Network Settings > Overview* in YaST and click *Edit*. The *Network Card Setup* dialog appears in which to adjust the card configuration using the *General*, *Address* and *Hardware* tabs. For information about wireless card configuration, see Section 24.5, « Configuration with YaST » (p. 423).

Configuring IP Addresses

You can set the IP address of the network card or the way its IP address is determined in the *Address* tab of the *Network Card Setup* dialog. Both IPv4 and IPv6 addresses are supported. The network card can have *No IP Address* (which is useful for bonding devices), a *Statically Assigned IP Address* (IPv4 or IPv6) or a *Dynamic Address* assigned via *DHCP* or *Zeroconf* or both.

If using *Dynamic Address*, select whether to use *DHCP Version 4 Only* (for IPv4), *DHCP Version 6 Only* (for IPv6) or *DHCP Both Version 4 and 6*.

If possible, the first network card with link that is available during the installation is automatically configured to use automatic address setup via DHCP. In case of laptop

computers where NetworkManager is active by default, all network cards are configured.

DHCP should also be used if you are using a DSL line but with no static IP assigned by the ISP (Internet Service Provider). If you decide to use DHCP, configure the details in *DHCP Client Options* in the *Global Options* tab of the *Network Settings* dialog of the YaST network card configuration module. If you have a virtual host setup where different hosts communicate through the same interface, an *DHCP Client Identifier* is necessary to distinguish them.

DHCP is a good choice for client configuration but it is not ideal for server configuration. To set a static IP address, proceed as follows:

- 1 Select a card from the list of detected cards in the *Overview* tab of the YaST network card configuration module and click *Edit*.
- 2 In the *Address* tab, choose *Statically Assigned IP Address*.
- 3 Enter the *IP Address*. Both IPv4 and IPv6 addresses can be used. Enter the network mask in *Subnet Mask*. If the IPv6 address is used, use *Subnet Mask* for prefix length in format / 64.

Optionally, you can enter a fully qualified *Hostname* for this address, which will be written to the `/etc/hosts` configuration file.

- 4 Click *Next*.
- 5 To activate the configuration, click *OK*.

If you use the static address, the name servers and default gateway are not configured automatically. To configure name servers, proceed as described in Section 13.4.1.4, « Configuring Hostname and DNS » (p. 245). To configure a gateway, proceed as described in Section 13.4.1.5, « Configuring Routing » (p. 247).

Configuring Aliases

One network device can have multiple IP addresses, called aliases.

REMARQUE : Aliases Are a Compatibility Feature

These so-called aliases resp. labels work with IPv4 only. With IPv6 they will be ignored. Using `iproute2` network interfaces can have one or more addresses.

Using YaST to set an alias for your network card, proceed as follows:

- 1 Select a card from the list of detected cards in the *Overview* tab of the YaST network card configuration module and click *Edit*.
- 2 In the *Address > Additional Addresses* tab, click *Add*.
- 3 Enter *Alias Name*, *IP Address*, and *Netmask*. Do not include the interface name in the alias name.
- 4 Click *OK*.
- 5 Click *Next*.
- 6 To activate the configuration, click *OK*.

Changing the Device Name and Udev Rules

It is possible to change the device name of the network card when it is used. It is also possible to determine whether the network card should be identified by udev via its hardware (MAC) address or via the bus ID. The later option is preferable in large servers to ease hot swapping of cards. To set these options with YaST, proceed as follows:

- 1 Select a card from the list of detected cards in the *Overview* tab of the YaST *Network Settings* module and click *Edit*.
- 2 Go to the *Hardware* tab. The current device name is shown in *Udev Rules*. Click *Change*.
- 3 Select whether udev should identify the card by its *MAC Address* or *Bus ID*. The current MAC address and bus ID of the card are shown in the dialog.
- 4 To change the device name, check the *Change Device Name* option and edit the name.

5 Click *OK* and *Next*.

6 To activate the configuration, click *OK*.

Changing Network Card Kernel Driver

For some network cards, several kernel drivers may be available. If the card is already configured, YaST allows you to select a kernel driver to be used from a list of available suitable drivers. It is also possible to specify options for the kernel driver. To set these options with YaST, proceed as follows:

- 1 Select a card from the list of detected cards in the *Overview* tab of the YaST Network Settings module and click *Edit*.
- 2 Go to the *Hardware* tab.
- 3 Select the kernel driver to be used in *Module Name*. Enter any options for the selected driver in *Options* in the form `option=value` . If more options are used, they should be space-separated.
- 4 Click *OK* and *Next*.
- 5 To activate the configuration, click *OK*.

Activating the Network Device

If you use the traditional method with `ifup`, you can configure your device to either start during boot, on cable connection, on card detection, manually or never. To change device start-up, proceed as follows:

- 1 In YaST select a card from the list of detected cards in *Network Devices > Network Settings* and click *Edit*.
- 2 In the *General* tab, select the desired entry from *Device Activation*.

Choose *At Boot Time* to start the device during the system boot. With *On Cable Connection*, the interface is watched for any existing physical connection. With *On Hotplug*, the interface is set as soon as available. It is similar to the *At Boot Time* option, and only differs in the fact that no error occurs if the interface is not present at boot time. Choose *Manually* to control the interface manually with

ifup. Choose *Never* to not start the device at all. The *On NFSroot* is similar to *At Boot Time*, but the interface does not shut down with the `rcnetwork stop` command. Use this if you use an nfs or iscsi root file system.

3 Click *Next*.

4 To activate the configuration, click *OK*.

Usually, only the system administrator can activate and deactivate network interfaces. If you want any user to be able to activate this interface via KInternet, select *Enable Device Control for Non-root User via KInternet*.

Setting Up Maximum Transfer Unit Size

You can set a maximum transmission unit (MTU) for the interface. MTU refers to the largest allowed packet size in bytes. A higher MTU brings higher bandwidth efficiency. However, large packets can block up a slow interface for some time, increasing the lag for further packets.

1 In YaST select a card from the list of detected cards in *Network Devices > Network Settings* and click *Edit*.

2 In the *General* tab, select the desired entry from the *Set MTU* list.

3 Click *Next*.

4 To activate the configuration, click *OK*.

Configuring the Firewall

Without having to enter the detailed firewall setup as described in section « Configuring the Firewall with YaST » (Chapitre 13, *Masquerading and Firewalls*, ↑Security Guide), you can determine the basic firewall setup for your device as part of the device setup. Proceed as follows:

1 Open the YaST *Network Devices > Network Settings* module. In the *Overview* tab, select a card from the list of detected cards and click *Edit*.

2 Enter the *General* tab of the *Network Settings* dialog.

- 3 Determine the firewall zone to which your interface should be assigned. The following options are available:

Firewall Disabled

This option is available only if the firewall is disabled and the firewall does not run at all. Only use this option if your machine is part of a greater network that is protected by an outer firewall.

Automatically Assign Zone

This option is available only if the firewall is enabled. The firewall is running and the interface is automatically assigned to a firewall zone. The zone which contains the keyword `any` or the external zone will be used for such an interface.

Internal Zone (Unprotected)

The firewall is running, but does not enforce any rules to protect this interface. Use this option if your machine is part of a greater network that is protected by an outer firewall. It is also useful for the interfaces connected to the internal network, when the machine has more network interfaces.

Demilitarized Zone

A demilitarized zone is an additional line of defense in front of an internal network and the (hostile) Internet. Hosts assigned to this zone can be reached from the internal network and from the Internet, but cannot access the internal network.

External Zone

The firewall is running on this interface and fully protects it against other—presumably hostile—network traffic. This is the default option.

- 4 Click *Next*.
- 5 Activate the configuration by clicking *OK*.

13.4.1.3 Configuring an Undetected Network Card

Your card may not be detected correctly. In this case, the card is not included in the list of detected cards. If you are sure that your system includes a driver for your card, you can configure it manually. You can also configure special network device types, such as bridge, bond, TUN or TAP. To configure an undetected network card (or a special device) proceed as follows:

- 1 In the *Network Devices > Network Settings > Overview* dialog in YaST click *Add*.
- 2 In the *Hardware* dialog, set the *Device Type* of the interface from the available options and *Configuration Name*. If the network card is a PCMCIA or USB device, activate the respective check box and exit this dialog with *Next*. Otherwise, you can define the kernel *Module Name* to be used for the card and its *Options*, if necessary.

In *Ethtool Options*, you can set `ethtool` options used by `ifup` for the interface. See the `ethtool` manual page for available options. If the option string starts with a `-` (for example `-K interface_name rx on`), the second word in the string is replaced with the current interface name. Otherwise (for example `autoneg off speed 10`) `ifup` prepends `-s interface_name`.

- 3 Click *Next*.
- 4 Configure any needed options, such as the IP address, device activation or fire-wall zone for the interface in the *General*, *Address*, and *Hardware* tabs. For more information about the configuration options, see Section 13.4.1.2, « Changing the Configuration of a Network Card » (p. 239).
- 5 If you selected *Wireless* as the device type of the interface, configure the wireless connection in the next dialog.
- 6 Click *Next*.
- 7 To activate the new network configuration, click *OK*.

13.4.1.4 Configuring Hostname and DNS

If you did not change the network configuration during installation and the wired card was already available, a hostname was automatically generated for your computer and DHCP was activated. The same applies to the name service information your host needs to integrate into a network environment. If DHCP is used for network address setup, the list of domain name servers is automatically filled with the appropriate data. If a static setup is preferred, set these values manually.

To change the name of your computer and adjust the name server search list, proceed as follows:

- 1 Go to the *Network Settings > Hostname/DNS* tab in the *Network Devices* module in YaST.

- 2 Enter the *Hostname* and, if needed, the *Domain Name*. The domain is especially important if the machine is a mail server. Note that the hostname is global and applies to all set network interfaces.

If you are using DHCP to get an IP address, the hostname of your computer will be automatically set by the DHCP. You may want to disable this behavior if you connect to different networks, because they may assign different hostnames and changing the hostname at runtime may confuse the graphical desktop. To disable using DHCP to get an IP address uncheck *Change Hostname via DHCP*.

Assign Hostname to Loopback IP associates your hostname with 127.0.0.2 (loopback) IP address in `/etc/hosts`. This is an useful option if you want to have the hostname resolvable at all times, even without active network.

- 3 In *Modify DNS Configuration*, select the way the DNS configuration (name servers, search list, the content of the `/etc/resolv.conf` file) is modified.

If the *Use Default Policy* option is selected, the configuration is handled by the `netconfig` script which merges the data defined statically (with YaST or in the configuration files) with data obtained dynamically (from the DHCP client or NetworkManager). This default policy is sufficient in most cases.

If the *Only Manually* option is selected, `netconfig` is not allowed to modify the `/etc/resolv.conf` file. However, this file can be edited manually.

If the *Custom Policy* option is selected, a *Custom Policy Rule* string defining the merge policy should be specified. The string consists of a comma-separated list of interface names to be considered a valid source of settings. Except for complete interface names, basic wildcards to match multiple interfaces are allowed, as well. For example, `eth* ppp?` will first target all `eth` and then all `ppp0-ppp9` interfaces. There are two special policy values that indicate how to apply the static settings defined in the `/etc/sysconfig/network/config` file:

STATIC

The static settings have to be merged together with the dynamic settings.

STATIC_FALLBACK

The static settings are used only when no dynamic configuration is available.

For more information, see the `man 8 netconfig`.

- 4 Enter the *Name Servers* and fill in the *Domain Search* list. Name servers must be specified by IP addresses, such as 192.168.1.116, not by hostnames. Names specified in the *Domain Search* tab are domain names used for resolving hostnames without a specified domain. If more than one *Domain Search* is used, separate domains with commas or white space.
- 5 To activate the configuration, click *OK*.

13.4.1.5 Configuring Routing

To make your machine communicate with other machines and other networks, routing information must be given to make network traffic take the correct path. If DHCP is used, this information is automatically provided. If a static setup is used, this data must be added manually.

- 1 In YaST go to *Network Settings > Routing*.
- 2 Enter the IP address of the *Default Gateway* (IPv4 and IPv6 if necessary). The default gateway matches every possible destination, but if any other entry exists that matches the required address, use this instead of the default route.
- 3 More entries can be entered in the *Routing Table*. Enter the *Destination* network IP address, *Gateway* IP address and the *Netmask*. Select the *Device* through which the traffic to the defined network will be routed (the minus sign stands for any device). To omit any of these values, use the minus sign -. To enter a default gateway into the table, use `default` in the *Destination* field.

REMARQUE

If more default routes are used, it is possible to specify the metric option to determine which route has a higher priority. To specify the metric option, enter `- metric number` in *Options*. The route with the highest metric is used as default. If the network device is disconnected, its route will be removed and the next one will be used. However, the current kernel does not use metric in static routing, only routing daemons like multipathd do.

- 4 If the system is a router, enable the *IP Forwarding* option in the *Network Settings*.
- 5 To activate the configuration, click *OK*.


13.4.2 Modem

In the YaST Control Center, access the modem configuration under *Network Devices* > *Modem*. If your modem was not automatically detected, go to the *Modem Devices* tab and open the dialog for manual configuration by clicking *Add*. Enter the interface to which the modem is connected under *Modem Device*.

ASTUCE : CDMA and GPRS Modems

Configure supported CDMA and GPRS modems with the YaST *Modem* module just as you would configure regular modems.

Figure 13.4 *Modem Configuration*

 **Modem Parameters**
Enter all modem configuration values. [more](#)

Modem Device:

Dial Prefix (if needed)

Dial Mode	Special Settings
<input checked="" type="radio"/> Tone Dialing	<input checked="" type="checkbox"/> Speaker On
<input type="radio"/> Pulse Dialing	<input checked="" type="checkbox"/> Detect Dial Tone

If you are behind a private branch exchange (PBX), you may need to enter a dial prefix. This is often a zero. Consult the instructions that came with the PBX to find out. Also select whether to use tone or pulse dialing, whether the speaker should be on and whether the modem should wait until it detects a dial tone. The last option should not be enabled if the modem is connected to an exchange.

Under *Details*, set the baud rate and the modem initialization strings. Only change these settings if your modem was not detected automatically or if it requires special settings for data transmission to work. This is mainly the case with ISDN terminal adapters. Leave this dialog by clicking *OK*. To delegate control over the modem to the normal user without root permissions, activate *Enable Device Control for Non-root User via KInternet*. In this way, a user without administrator permissions can activate or deactivate an interface. Under *Dial Prefix Regular Expression*, specify a regular expression. The *Dial Prefix* in KInternet, which can be modified by the normal user, must match this regular expression. If this field is left empty, the user cannot set a different *Dial Prefix* without administrator permissions.

In the next dialog, select the ISP. To choose from a predefined list of ISPs operating in your country, select *Country*. Alternatively, click *New* to open a dialog in which to provide the data for your ISP. This includes a name for the dial-up connection and ISP as well as the login and password provided by your ISP. Enable *Always Ask for Password* to be prompted for the password each time you connect.

In the last dialog, specify additional connection options:

Dial on Demand

If you enable *Dial on Demand*, set at least one name server. Use this feature only if your Internet connection is inexpensive, because there are programs that periodically request data from the Internet.

Modify DNS when Connected

This option is enabled by default, with the effect that the name server address is updated each time you connect to the Internet.

Automatically Retrieve DNS

If the provider does not transmit its domain name server after connecting, disable this option and enter the DNS data manually.

Automatically Reconnect

If this options is enabled, the connection is automatically reestablished after failure.

Ignore Prompts

This option disables the detection of any prompts from the dial-up server. If the connection build-up is slow or does not work at all, try this option.

External Firewall Interface

Selecting this option activates the firewall and sets the interface as external. This way, you are protected from outside attacks for the duration of your Internet connection.

Idle Time-Out (seconds)

With this option, specify a period of network inactivity after which the modem disconnects automatically.

IP Details

This opens the address configuration dialog. If your ISP does not assign a dynamic IP address to your host, disable *Dynamic IP Address* then enter your host's local IP address and the remote IP address. Ask your ISP for this information. Leave *Default Route* enabled and close the dialog by selecting *OK*.

Selecting *Next* returns to the original dialog, which displays a summary of the modem configuration. Close this dialog with *OK*.

13.4.3 ISDN

Use this module to configure one or several ISDN cards for your system. If YaST did not detect your ISDN card, click on *Add* in the *ISDN Devices* tab and manually select your card. Multiple interfaces are possible, but several ISPs can be configured for one interface. In the subsequent dialogs, set the ISDN options necessary for the proper functioning of the card.

Figure 13.5 ISDN Configuration

ISDN Low-Level Configuration for contr0
With OnBoot, the driver is loaded during system boot. [more](#)

ISDN Card Information

Vendor	Abocom/Magitek
ISDN Card	2BD1

Driver:

ISDN Protocol

☒ Euro-ISDN (EDSSI)
☐ 1TR6
☐ Leased Line
☐ NI1

Country:

Code:

Area Code:

Dial Prefix:

☒ Start ISDN Log

Activate device:

In the next dialog, shown in Figure 13.5, « ISDN Configuration » (p. 251), select the protocol to use. The default is *Euro-ISDN (EDSSI)*, but for older or larger exchanges, select *1TR6*. If you are in the US, select *NI1*. Select your country in the relevant field. The corresponding country code then appears in the field next to it. Finally, provide your *Area Code* and the *Dial Prefix* if necessary. If you do not want to log all your ISDN traffic, uncheck the *Start ISDN Log* option.

Activate Device defines how the ISDN interface should be started: *At Boot Time* causes the ISDN driver to be initialized each time the system boots. *Manually* requires you to load the ISDN driver as root with the command `rcisdn start`. *On Hotplug*, used for PCMCIA or USB devices, loads the driver after the device is plugged in. When finished with these settings, select *OK*.

In the next dialog, specify the interface type for your ISDN card and add ISPs to an existing interface. Interfaces may be either the *SyncPPP* or the *RawIP* type, but most ISPs operate in the *SyncPPP* mode, which is described below.

Figure 13.6 ISDN Interface Configuration

The number to enter for *My Phone Number* depends on your particular setup:

ISDN Card Directly Connected to Phone Outlet

A standard ISDN line provides three phone numbers (called multiple subscriber numbers, or MSNs). If the subscriber asked for more, there may be up to 10. One of these MSNs must be entered here, but without your area code. If you enter the wrong number, your phone operator automatically falls back to the first MSN assigned to your ISDN line.

ISDN Card Connected to a Private Branch Exchange

Again, the configuration may vary depending on the equipment installed:

1. Smaller private branch exchanges (PBX) built for home purposes mostly use the Euro-ISDN (EDSS1) protocol for internal calls. These exchanges have an internal S0 bus and use internal numbers for the equipment connected to them.

Use one of the internal numbers as your MSN. You should be able to use at least one of the exchange's MSNs that have been enabled for direct outward dialing. If this does not work, try a single zero. For further information, consult the documentation delivered with your phone exchange.

2. Larger phone exchanges designed for businesses normally use the ITR6 protocol for internal calls. Their MSN is called EAZ and usually corresponds to the direct-dial number. For the configuration under Linux, it should be sufficient to enter the last digit of the EAZ. As a last resort, try each of the digits from 1 to 9.

For the connection to be terminated just before the next charge unit is due, enable *ChargeHUP*. However, remember that may not work with every ISP. You can also enable channel bundling (multilink PPP) by selecting the corresponding option. Finally, you can enable the firewall for your link by selecting *External Firewall Interface* and *Restart Firewall*. To enable the normal user without administrator permissions to activate or deactivate the interface, select the *Enable Device Control for Non-root User via KInternet*.

Details opens a dialog in which to implement more complex connection schemes which are not relevant for normal home users. Leave the *Details* dialog by selecting *OK*.

In the next dialog, configure IP address settings. If you have not been given a static IP by your provider, select *Dynamic IP Address*. Otherwise, use the fields provided to enter your host's local IP address and the remote IP address according to the specifications of your ISP. If the interface should be the default route to the Internet, select *Default Route*. Each host can only have one interface configured as the default route. Leave this dialog by selecting *Next*.

The following dialog allows you to set your country and select an ISP. The ISPs included in the list are call-by-call providers only. If your ISP is not in the list, select *New*. This opens the *Provider Parameters* dialog in which to enter all the details for your ISP. When entering the phone number, do not include any blanks or commas among the digits. Finally, enter your login and the password as provided by the ISP. When finished, select *Next*.

To use *Dial on Demand* on a stand-alone workstation, specify the name server (DNS server) as well. Most ISPs support dynamic DNS, which means the IP address of a name server is sent by the ISP each time you connect. For a single workstation, however, you still need to provide a placeholder address like 192.168.22.99. If your ISP does not support dynamic DNS, specify the name server IP addresses of the ISP. If desired, specify a time-out for the connection—the period of network inactivity (in seconds) after which the connection should be automatically terminated. Confirm your settings with *Next*. YaST displays a summary of the configured interfaces. To activate these settings, select *OK*.

13.4.4 Cable Modem

In some countries it is quite common to access the Internet through the TV cable network. The TV cable subscriber usually gets a modem that is connected to the TV cable outlet on one side and to a computer network card on the other (using a 10Base-TG twisted pair cable). The cable modem then provides a dedicated Internet connection with a fixed IP address.

Depending on the instructions provided by your ISP, when configuring the network card either select *Dynamic Address* or *Statically Assigned IP Address*. Most providers today use DHCP. A static IP address often comes as part of a special business account.

13.4.5 DSL

To configure your DSL device, select the *DSL* module from the YaST *Network Devices* section. This YaST module consists of several dialogs in which to set the parameters of DSL links based on one of the following protocols:

- PPP over Ethernet (PPPoE)
- PPP over ATM (PPPoATM)
- CAPI for ADSL (Fritz Cards)
- Point-to-Point Tunneling Protocol (PPTP)—Austria

In the *DSL Devices* tab of the *DSL Configuration Overview* dialog, you will find a list of installed DSL devices. To change the configuration of a DSL device, select it in the list and click *Edit*. If you click *Add*, you can manually configure a new DSL device.

The configuration of a DSL connection based on PPPoE or PPTP requires that the corresponding network card be set up in the correct way. If you have not done so yet, first configure the card by selecting *Configure Network Cards* (see Section 13.4.1, « Configuring the Network Card with YaST » (p. 237)). In the case of a DSL link, addresses may be assigned automatically but not via DHCP, which is why you should not enable the option *Dynamic Address*. Instead, enter a static dummy address for the interface, such as 192.168.22.1. In *Subnet Mask*, enter 255.255.255.0. If you are configuring a stand-alone workstation, leave *Default Gateway* empty.


ASTUCE

Values in *IP Address* and *Subnet Mask* are only placeholders. They are only needed to initialize the network card and do not represent the DSL link as such.

In the first DSL configuration dialog (see Figure 13.7, « DSL Configuration » (p. 256)), select the *PPP Mode* and the *Ethernet Card* to which the DSL modem is connected (in most cases, this is `eth0`). Then use *Activate Device* to specify whether the DSL link should be established during the boot process. Click *Enable Device Control for Non-root User via KInternet* to authorize the normal user without root permissions to activate or deactivate the interface with KInternet.

In the next dialog select your country and choose from a number of ISPs operating in it. The details of any subsequent dialogs of the DSL configuration depend on the options set so far, which is why they are only briefly mentioned in the following paragraphs. For details on the available options, read the detailed help available from the dialogs.

Figure 13.7 DSL Configuration

 **DSL Configuration**
Here, set the most important settings for the DSL connection. [more](#)

DSL Connection Settings

PPP Mode:
PPP over Ethernet

PPP Mode-Dependent Settings

VPI/VCI:

Ethernet Card

82566DM Gigabit Network Connection
Network Card - DHCP address

Server Name or IP Address:

Activate device:
Manually

☒ Enable Device Control for Non-root User Via KInetnet

To use *Dial on Demand* on a stand-alone workstation, also specify the name server (DNS server). Most ISPs support dynamic DNS—the IP address of a name server is sent by the ISP each time you connect. For a single workstation, however, provide a placeholder address like 192.168.22.99. If your ISP does not support dynamic DNS, enter the name server IP address provided by your ISP.

Idle Time-Out (seconds) defines a period of network inactivity after which to terminate the connection automatically. A reasonable time-out value is between 60 and 300 seconds. If *Dial on Demand* is disabled, it may be useful to set the time-out to zero to prevent automatic hang-up.

The configuration of T-DSL is very similar to the DSL setup. Just select *T-Online* as your provider and YaST opens the T-DSL configuration dialog. In this dialog, provide some additional information required for T-DSL—the line ID, the T-Online number, the user code and your password. All of these should be included in the information you received after subscribing to T-DSL.

13.5 NetworkManager

NetworkManager is the ideal solution for laptops and other portable computers. With NetworkManager, you do not need to worry about configuring network interfaces and switching between networks when you are moving.

13.5.1 NetworkManager and ifup

However, NetworkManager is not a suitable solution for all cases, so you can still choose between the traditional method for managing network connections (ifup) and NetworkManager. If you want to manage your network connection with NetworkManager, enable NetworkManager in the YaST Network Settings module as described in Section 25.2, « Enabling or Disabling NetworkManager » (p. 438) and configure your network connections with NetworkManager. For a list of use cases and a detailed description how to configure and use NetworkManager, refer to Chapitre 25, *Using NetworkManager* (p. 437).

Some differences between ifup and NetworkManager include:

`root` Privileges

If you use NetworkManager for network setup, you can easily switch, stop or start your network connection at any time from an icon within your desktop environment. NetworkManager also makes it possible to change and configure wireless card connections without requiring `root` privileges. For this reason, NetworkManager is the ideal solution for a mobile workstation.

Traditional configuration with ifup also provides some ways to switch, stop or start the connection with or without user intervention, like user-managed devices. However, this always requires `root` privileges to change or configure a network device. This is often a problem for mobile computing, where it is not possible to preconfigure all the connection possibilities.

Types of Network Connections

Both traditional configuration and NetworkManager can handle network connections with a wireless network (with WEP, WPA-PSK, and WPA-Enterprise access) and wired networks using DHCP and static configuration. They also support connection through dial-up, DSL and VPN. With NetworkManager you can also connect a mobile broadband (3G) modem, which is not possible with the traditional configuration.

NetworkManager tries to keep your computer connected at all times using the best connection available. If the network cable is accidentally disconnected, it tries to reconnect. It can find the network with the best signal strength from the list of your wireless connections and automatically use it to connect. To get the same functionality with `ifup`, a great deal of configuration effort is required.

13.5.2 NetworkManager Functionality and Configuration Files

The individual network connection settings created with NetworkManager are stored in configuration profiles. The *system* connections configured with either NetworkManager or YaST are saved in `/etc/networkmanager/system-connections/*` or in `/etc/sysconfig/network/ifcfg-*`. Any user-defined connections are stored in GConf for GNOME.

In case no profile is configured, NetworkManager automatically creates one and names it `Auto $INTERFACE-NAME`. That is made in an attempt to work without any configuration for as many cases as (securely) possible. If the automatically created profiles do not suit your needs, use the network connection configuration dialogs provided by KDE or GNOME to modify them as desired. For more information, refer to Section 25.3, « Configuring Network Connections » (p. 439).

13.5.3 Controlling and Locking Down NetworkManager Features

On centrally administered machines, certain NetworkManager features can be controlled or disabled with PolicyKit, for example if a user is allowed to modify administrator defined connections or if a user is allowed to define his own network configurations. To view or change the respective NetworkManager policies, start the graphical *Authorizations* tool for PolicyKit. In the tree on the left side, find them below the *network-manager-settings* entry.

The following table gives an overview of the PolicyKit identifiers related to NetworkManager:

Tableau 13.5 *PolicyKit Identifiers for NetworkManager*

Identifier	Description
org.freedesktop.NetworkManager.enable-disable-network	Enable or disable system networking.
org.freedesktop.NetworkManager.sleep-wake	Put NetworkManager to sleep or wake it up.
org.freedesktop.NetworkManager.enable-disable-wwan	Enable or disable mobile broadband devices.
org.freedesktop.NetworkManager.enable-disable-wimax	Enable or disable WiMAX mobile broadband devices.
org.freedesktop.NetworkManager.network-control	Allow control of network connections.
org.freedesktop.NetworkManager.enable-disable-wifi	Enable or disable WiFi devices.
org.freedesktop.NetworkManager.settings.modify.hosts	Modify persistent system hostname.
org.freedesktop.network-manager-settings.modify.system	Modify network connections for all users.
org.freedesktop.network-manager-settings.modify.own	Modify personal network connections.
org.freedesktop.network-manager-settings.system.wifi.share.open	Connection sharing via an open WiFi network.
org.freedesktop.network-manager-settings.system.wifi.share.protected	Connection sharing via a protected WiFi network.

13.6 Configuring a Network Connection Manually

Manual configuration of the network software should always be the last alternative. Using YaST is recommended. However, this background information about the network configuration can also assist your work with YaST.

When the Kernel detects a network card and creates a corresponding network interface, it assigns the device a name depending on the order of device discovery, or order of the loading of the Kernel modules. The default Kernel device names are only predictable in very simple or tightly controlled hardware environments. Systems which allow adding or removing hardware during runtime or support automatic configuration of devices cannot expect stable network device names assigned by the Kernel across reboots.

However, all system configuration tools rely on persistent interface names. This problem is solved by udev. The udev persistent net generator (`/lib/udev/rules.d/75-persistent-net-generator.rules`) generates a rule matching the hardware (using its hardware address by default) and assigns a persistently unique interface for the hardware. The udev database of network interfaces is stored in the file `/etc/udev/rules.d/70-persistent-net.rules`. Every line in the file describes one network interface and specifies its persistent name. System administrators can change the assigned names by editing the `NAME=""` entries. The persistent rules can also be modified using YaST.

Tableau 13.6, « Manual Network Configuration Scripts » (p. 260) summarizes the most important scripts involved in the network configuration.

Tableau 13.6 *Manual Network Configuration Scripts*

Command	Function
<code>ifup, ifdown, ifstatus</code>	The <code>if</code> scripts start or stop network interfaces, or return the status of the specified interface. For more information, see the <code>ifup</code> manual page.
<code>rcnetwork</code>	The <code>rcnetwork</code> script can be used to start, stop or restart all network

Command	Function
	interfaces (or just a specified one). Use <code>rcnetwork stop</code> to stop, <code>rcnetwork start</code> to start and <code>rcnetwork restart</code> to restart network interfaces. If you want to stop, start or restart just one interface, use the command followed by the interface name, for example <code>rcnetwork restart eth0</code> . The <code>rcnetwork status</code> command displays the state of the interfaces, their IP addresses and whether a DHCP client is running. With <code>rcnetwork stop-all- dhcp-clients</code> and <code>rcnetwork restart-all-dhcp-clients</code> you can stop or restart DHCP clients running on network interfaces.

For more information about udev and persistent device names, see Chapitre 12, *Dynamic Kernel Device Management with udev* (p. 203).

13.6.1 Configuration Files

This section provides an overview of the network configuration files and explains their purpose and the format used.

13.6.1.1 /etc/sysconfig/network/ifcfg-*

These files contain the configurations for network interfaces. They include information such as the start mode and the IP address. Possible parameters are described in the manual page of `ifup`. Additionally, most variables from the `dhcp` and `wireless` files can be used in the `ifcfg-*` files if a general setting should be used for only one interface. However, most of the `/etc/sysconfig/network/config` variables are global and cannot be overridden in `ifcfg`-files. For example `NETWORK-MANAGER` or `NETCONFIG_*` variables are global.

For `ifcfg.template`, see Section 13.6.1.2, « `/etc/sysconfig/network/config`, `/etc/sysconfig/network/dhcp`, and `/etc/sysconfig/network/wireless` » (p. 262).

13.6.1.2 `/etc/sysconfig/network/config`, `/etc/sysconfig/network/dhcp`, and `/etc/sysconfig/network/wireless`

The file `config` contains general settings for the behavior of `ifup`, `ifdown` and `ifstatus`. `dhcp` contains settings for DHCP and `wireless` for wireless LAN cards. The variables in all three configuration files are commented. Some of the variables from `/etc/sysconfig/network/config` can also be used in `ifcfg-*` files, where they are given a higher priority. The `/etc/sysconfig/network/ifcfg.template` file lists variables that can be specified in a per interface scope. However, most of the `/etc/sysconfig/network/config` variables are global and cannot be overridden in `ifcfg`-files. For example, `NETWORK-MANAGER` or `NETCONFIG_*` variables are global.

13.6.1.3 `/etc/sysconfig/network/routes` and `/etc/sysconfig/network/ifroute-*`

The static routing of TCP/IP packets is determined here. All the static routes required by the various system tasks can be entered in the `/etc/sysconfig/network/routes` file: routes to a host, routes to a host via a gateway and routes to a network. For each interface that needs individual routing, define an additional configuration file: `/etc/sysconfig/network/ifroute-*`. Replace `*` with the name of the interface. The entries in the routing configuration files look like this:

# Destination	Dummy/Gateway	Netmask	Device
#			
127.0.0.0	0.0.0.0	255.255.255.0	lo
204.127.235.0	0.0.0.0	255.255.255.0	eth0
default	204.127.235.41	0.0.0.0	eth0
207.68.156.51	207.68.145.45	255.255.255.255	eth1
192.168.0.0	207.68.156.51	255.255.0.0	eth1

The route's destination is in the first column. This column may contain the IP address of a network or host or, in the case of *reachable* name servers, the fully qualified network or hostname.

The second column contains the default gateway or a gateway through which a host or network can be accessed. The third column contains the netmask for networks or

hosts behind a gateway. For example, the mask is 255.255.255.255 for a host behind a gateway.

The fourth column is only relevant for networks connected to the local host such as loopback, Ethernet, ISDN, PPP and dummy device. The device name must be entered here.

An (optional) fifth column can be used to specify the type of a route. Columns that are not needed should contain a minus sign – to ensure that the parser correctly interprets the command. For details, refer to the `routes(5)` man page.

The unified format for IPv4 and IPv6 now looks as follows:

```
prefix/length      gateway -           [interface]
```

And the so-called compatibility format looks accordingly:

```
prefix      gateway length      [interface]
```

For IPv4 you still can use the old format with netmask:

```
ipv4-network      gateway ipv4-netmask [interface]
```

The following examples are equivalent:

2001:db8:abba:cafe::/64	2001:db8:abba:cafe::dead	-	eth0
208.77.188.0/24	208.77.188.166	-	eth0
2001:db8:abba:cafe::	2001:db8:abba:cafe::dead	64	eth0
208.77.188.0	208.77.188.166	24	eth0
208.77.188.0	208.77.188.166	255.255.255.0	eth0

13.6.1.4 /etc/resolv.conf

The domain to which the host belongs is specified in this file (keyword search). Also listed is the status of the name server address to access (keyword name-server). Multiple domain names can be specified in the file. When resolving a name that is not fully qualified, an attempt is made to generate one by attaching the individual search entries. Multiple name servers can be specified in multiple lines, each beginning with `nameserver`. Comments are preceded by `#` signs. Example 13.5, « `/etc/resolv.conf` » (p. 264) shows what `/etc/resolv.conf` could look like.

However, the `/etc/resolv.conf` should not be edited by hand. Instead, it is generated by the `netconfig` script. To define static DNS configuration without

using YaST, edit the appropriate variables manually in the `/etc/sysconfig/network/config` file:

`NETCONFIG_DNS_STATIC_SEARCHLIST`
list of DNS domain names used for hostname lookup

`NETCONFIG_DNS_STATIC_SERVERS`
list of name server IP addresses to use for hostname lookup

`NETCONFIG_DNS_FORWARDER`
defines the name of the DNS forwarder that has to be configured

To disable DNS configuration using `netconfig`, set `NETCONFIG_DNS_POLICY=''`. For more information about `netconfig`, see `man 8 netconfig`.

Example 13.5 */etc/resolv.conf*

```
# Our domain
search example.com
#
# We use dns.example.com (192.168.1.116) as nameserver
nameserver 192.168.1.116
```

13.6.1.5 /sbin/netconfig

`netconfig` is a modular tool to manage additional network configuration settings. It merges statically defined settings with settings provided by autoconfiguration mechanisms as DHCP or PPP according to a predefined policy. The required changes are applied to the system by calling the `netconfig` modules that are responsible for modifying a configuration file and restarting a service or a similar action.

`netconfig` recognizes three main actions. The `netconfig modify` and `netconfig remove` commands are used by daemons such as DHCP or PPP to provide or remove settings to `netconfig`. Only the `netconfig update` command is available for the user:

`modify`

The `netconfig modify` command modifies the current interface and service specific dynamic settings and updates the network configuration. `Netconfig` reads settings from standard input or from a file specified with the `--lease-file filename` option and internally stores them until a system reboot (or the next `modify` or `remove` action). Already existing settings for the same interface and service combination are overwritten. The interface is specified by

the `-i interface_name` parameter. The service is specified by the `-s service_name` parameter.

remove

The `netconfig remove` command removes the dynamic settings provided by a modificatory action for the specified interface and service combination and updates the network configuration. The interface is specified by the `-i interface_name` parameter. The service is specified by the `-s service_name` parameter.

update

The `netconfig update` command updates the network configuration using current settings. This is useful when the policy or the static configuration has changed. Use the `-m module_type` parameter, if you want to update a specified service only (dns, nis, or ntp).

The `netconfig` policy and the static configuration settings are defined either manually or using YaST in the `/etc/sysconfig/network/config` file. The dynamic configuration settings provided by autoconfiguration tools as DHCP or PPP are delivered directly by these tools with the `netconfig modify` and `netconfig remove` actions. NetworkManager also uses `netconfig modify` and `netconfig remove` actions. When NetworkManager is enabled, `netconfig` (in policy mode `auto`) uses only NetworkManager settings, ignoring settings from any other interfaces configured using the traditional `ifup` method. If NetworkManager does not provide any setting, static settings are used as a fallback. A mixed usage of NetworkManager and the traditional `ifup` method is not supported.

For more information about `netconfig`, see `man 8 netconfig`.

13.6.1.6 /etc/hosts

In this file, shown in Example 13.6, « `/etc/hosts` » (p. 265), IP addresses are assigned to hostnames. If no name server is implemented, all hosts to which an IP connection will be set up must be listed here. For each host, enter a line consisting of the IP address, the fully qualified hostname, and the hostname into the file. The IP address must be at the beginning of the line and the entries separated by blanks and tabs. Comments are always preceded by the `#` sign.

Example 13.6 `/etc/hosts`

```
127.0.0.1 localhost
```

```
192.168.2.100 jupiter.example.com jupiter
192.168.2.101 venus.example.com venus
```

13.6.1.7 /etc/networks

Here, network names are converted to network addresses. The format is similar to that of the `hosts` file, except the network names precede the addresses. See Example 13.7, « `/etc/networks` » (p. 266).

Exemple 13.7 */etc/networks*

```
loopback      127.0.0.0
localnet      192.168.0.0
```

13.6.1.8 /etc/host.conf

Name resolution—the translation of host and network names via the *resolver* library—is controlled by this file. This file is only used for programs linked to `libc4` or `libc5`. For current `glibc` programs, refer to the settings in `/etc/nsswitch.conf`. A parameter must always stand alone in its own line. Comments are preceded by a `#` sign. Tableau 13.7, « Parameters for `/etc/host.conf` » (p. 266) shows the parameters available. A sample `/etc/host.conf` is shown in Exemple 13.8, « `/etc/host.conf` » (p. 267).

Tableau 13.7 *Parameters for /etc/host.conf*

<code>order hosts, bind</code>	Specifies in which order the services are accessed for the name resolution. Available arguments are (separated by blank spaces or commas):
	<code>hosts</code> : searches the <code>/etc/hosts</code> file
	<code>bind</code> : accesses a name server
	<code>nis</code> : uses NIS
<code>multi on/off</code>	Defines if a host entered in <code>/etc/hosts</code> can have multiple IP addresses.

<code>nospoof on spoofalert on/off</code>	These parameters influence the name server <i>spoofing</i> but do not exert any influence on the network configuration.
<code>trim domainname</code>	The specified domain name is separated from the hostname after host-name resolution (as long as the host-name includes the domain name). This option is useful only if names from the local domain are in the <code>/etc/hosts</code> file, but should still be recognized with the attached domain names.

Example 13.8 */etc/host.conf*

```
# We have named running
order hosts bind
# Allow multiple address
multi on
```

13.6.1.9 /etc/nsswitch.conf

The introduction of the GNU C Library 2.0 was accompanied by the introduction of the *Name Service Switch* (NSS). Refer to the `nsswitch.conf(5)` man page and *The GNU C Library Reference Manual* for details.

The order for queries is defined in the file `/etc/nsswitch.conf`. A sample `nsswitch.conf` is shown in Example 13.9, « `/etc/nsswitch.conf` » (p. 267). Comments are preceded by `#` signs. In this example, the entry under the `hosts` database means that a request is sent to `/etc/hosts (files)` via DNS.

Example 13.9 */etc/nsswitch.conf*

```
passwd:      compat
group:       compat

hosts:       files dns
networks:    files dns
```

```

services:  db files
protocols: db files
rpc:       files
ethers:    files
netmasks: files
netgroup:  files nis
publickey: files

bootparams: files
automount:  files nis
aliases:    files nis
shadow:     compat

```

The « databases » available over NSS are listed in Tableau 13.8, « Databases Available via /etc/nsswitch.conf » (p. 268). The configuration options for NSS databases are listed in Tableau 13.9, « Configuration Options for NSS « Databases » » (p. 269).

Tableau 13.8 *Databases Available via /etc/nsswitch.conf*

aliases	Mail aliases implemented by send mail; see man 5 aliases.
ethers	Ethernet addresses.
netmasks	List of network and their subnet masks. Only needed, if you use subnetting.
group	For user groups used by getgrent. See also the man page for group.
hosts	For hostnames and IP addresses, used by gethostbyname and similar functions.
netgroup	Valid host and user lists in the network for the purpose of controlling access permissions; see the netgroup (5) man page.
networks	Network names and addresses, used by getnetent.

publickey	Public and secret keys for Secure_RPC used by NFS and NIS+..
passwd	User passwords, used by getpwent; see the passwd (5) man page.
protocols	Network protocols, used by getprotoent; see the protocols (5) man page.
rpc	Remote procedure call names and addresses, used by getrpcbyname and similar functions.
services	Network services, used by getservent.
shadow	Shadow passwords of users, used by getspnam; see the shadow (5) man page.

Tableau 13.9 *Configuration Options for NSS « Databases »*

files	directly access files, for example, /etc/aliases
db	access via a database
nis,nisplus	NIS, see also Chapitre 3, <i>Using NIS</i> (↑Security Guide)
dns	can only be used as an extension for hosts and networks
compat	can only be used as an extension for passwd, shadow and group

13.6.1.10 /etc/nscd.conf

This file is used to configure `nsd` (name service cache daemon). See the `nsd(8)` and `nsd.conf(5)` man pages. By default, the system entries of `passwd` and `groups` are cached by `nsd`. This is important for the performance of directory services, like NIS and LDAP, because otherwise the network connection needs to be used for every access to names or groups. `hosts` is not cached by default, because the mechanism in `nsd` to cache `hosts` makes the local system unable to trust forward and reverse lookup checks. Instead of asking `nsd` to cache names, set up a caching DNS server.

If the caching for `passwd` is activated, it usually takes about fifteen seconds until a newly added local user is recognized. Reduce this waiting time by restarting `nsd` with the command `rcnsd restart`.

13.6.1.11 /etc/HOSTNAME

This contains the fully qualified hostname with the domain name attached. This file is read by several scripts while the machine is booting. It must contain only one line (in which the hostname is set).

13.6.2 Testing the Configuration

Before you write your configuration to the configuration files, you can test it. To set up a test configuration, use the `ip` command. To test the connection, use the `ping` command. Older configuration tools, `ifconfig` and `route`, are also available.

The commands `ip`, `ifconfig` and `route` change the network configuration directly without saving it in the configuration file. Unless you enter your configuration in the correct configuration files, the changed network configuration is lost on reboot.

13.6.2.1 Configuring a Network Interface with `ip`

`ip` is a tool to show and configure network devices, routing, policy routing, and tunnels.

`ip` is a very complex tool. Its common syntax is `ip options object command`. You can work with the following objects:

link

This object represents a network device.

address

This object represents the IP address of device.

neighbor

This object represents a ARP or NDISC cache entry.

route

This object represents the routing table entry.

rule

This object represents a rule in the routing policy database.

maddress

This object represents a multicast address.

mroute

This object represents a multicast routing cache entry.

tunnel

This object represents a tunnel over IP.

If no command is given, the default command is used (usually `list`).

Change the state of a device with the command `ip link`

`set device_name command`. For example, to deactivate device `eth0`, enter `ip link set eth0 down`. To activate it again, use `ip link set eth0 up`.

After activating a device, you can configure it. To set the IP address, use `ip addr add ip_address + dev device_name`. For example, to set the address of the interface `eth0` to `192.168.12.154/30` with standard broadcast (option `brd`), enter `ip addr add 192.168.12.154/30 brd + dev eth0`.

To have a working connection, you must also configure the default gateway. To set a gateway for your system, enter `ip route add gateway_ip_address`.

To translate one IP address to another, use `nat: ip route add nat_ip_address via other_ip_address`.

To display all devices, use `ip link ls`. To display the running interfaces only, use `ip link ls up`. To print interface statistics for a device, enter `ip -s link ls device_name`. To view addresses of your devices, enter `ip addr`. In the

output of the `ip addr`, also find information about MAC addresses of your devices. To show all routes, use `ip route show`.

For more information about using `ip`, enter `ip help` or see the `ip(8)` man page. The `help` option is also available for all `ip` subcommands. If, for example, you need help for `ip addr`, enter `ip addr help`. Find the `ip` manual in `/usr/share/doc/packages/iproute2/ip-cref.pdf`.

13.6.2.2 Testing a Connection with ping

The `ping` command is the standard tool for testing whether a TCP/IP connection works. It uses the ICMP protocol to send a small data packet, `ECHO_REQUEST` datagram, to the destination host, requesting an immediate reply. If this works, `ping` displays a message to that effect, which indicates that the network link is basically functioning.

`ping` does more than only test the function of the connection between two computers: it also provides some basic information about the quality of the connection. In Example 13.10, « Output of the Command `ping` » (p. 272), you can see an example of the `ping` output. The second-to-last line contains information about the number of transmitted packets, packet loss, and total time of `ping` running.

As the destination, you can use a hostname or IP address, for example, `ping example.com` or `ping 192.168.3.100`. The program sends packets until you press `Ctrl + C`.

If you only need to check the functionality of the connection, you can limit the number of the packets with the `-c` option. For example to limit `ping` to three packets, enter `ping -c 3 example.com`.

Example 13.10 *Output of the Command `ping`*

```
ping -c 3 example.com
PING example.com (192.168.3.100) 56(84) bytes of data.
64 bytes from example.com (192.168.3.100): icmp_seq=1 ttl=49 time=188 ms
64 bytes from example.com (192.168.3.100): icmp_seq=2 ttl=49 time=184 ms
64 bytes from example.com (192.168.3.100): icmp_seq=3 ttl=49 time=183 ms
--- example.com ping statistics ---
3 packets transmitted, 3 received, 0% packet loss, time 2007ms
rtt min/avg/max/mdev = 183.417/185.447/188.259/2.052 ms
```

The default interval between two packets is one second. To change the interval, `ping` provides the option `-i`. For example, to increase the `ping` interval to ten seconds, enter `ping -i 10 example.com`.

In a system with multiple network devices, it is sometimes useful to send the ping through a specific interface address. To do so, use the `-I` option with the name of the selected device, for example, `ping -I wlan1 example.com`.

For more options and information about using ping, enter `ping -h` or see the ping (8) man page.

ASTUCE : Pinging IPv6 Addresses

For IPv6 addresses use the `ping6` command. Note, to ping link-local addresses, you must specify the interface with `-I`. The following command works, if the address is reachable via `eth1`:

```
ping6 -I eth1 fe80::117:21ff:feda:a425
```

13.6.2.3 Configuring the Network with ifconfig

`ifconfig` is a network configuration tool.

REMARQUE : `ifconfig` and `ip`

The `ifconfig` tool is obsolete. Use `ip` instead. In contrast to `ip`, you can use `ifconfig` only for interface configuration. It limits interface names to 9 characters.

Without arguments, `ifconfig` displays the status of the currently active interfaces. As you can see in Exemple 13.11, « Output of the `ifconfig` Command » (p. 273), `ifconfig` has very well-arranged and detailed output. The output also contains information about the MAC address of your device (the value of `HWaddr`) in the first line.

Exemple 13.11 *Output of the ifconfig Command*

```
eth0      Link encap:Ethernet  HWaddr 00:08:74:98:ED:51
          inet6 addr: fe80::208:74ff:fe98:ed51/64 Scope:Link
          UP BROADCAST MULTICAST  MTU:1500  Metric:1
          RX packets:634735 errors:0 dropped:0 overruns:4 frame:0
          TX packets:154779 errors:0 dropped:0 overruns:0 carrier:1
          collisions:0 txqueuelen:1000
          RX bytes:162531992 (155.0 Mb)  TX bytes:49575995 (47.2 Mb)
          Interrupt:11 Base address:0xec80

lo        Link encap:Local Loopback
```

```

    inet addr:127.0.0.1  Mask:255.0.0.0
    inet6 addr: ::1/128 Scope:Host
    UP LOOPBACK RUNNING  MTU:16436  Metric:1
    RX packets:8559 errors:0 dropped:0 overruns:0 frame:0
    TX packets:8559 errors:0 dropped:0 overruns:0 carrier:0
    collisions:0 txqueuelen:0
    RX bytes:533234 (520.7 Kb)  TX bytes:533234 (520.7 Kb)

wlan1    Link encap:Ethernet  HWaddr 00:0E:2E:52:3B:1D
    inet addr:192.168.2.4  Bcast:192.168.2.255  Mask:255.255.255.0
    inet6 addr: fe80::20e:2eff:fe52:3b1d/64  Scope:Link
    UP BROADCAST NOTRAILERS RUNNING MULTICAST  MTU:1500  Metric:1
    RX packets:50828 errors:0 dropped:0 overruns:0 frame:0
    TX packets:43770 errors:0 dropped:0 overruns:0 carrier:0
    collisions:0 txqueuelen:1000
    RX bytes:45978185 (43.8 Mb)  TX bytes:7526693 (7.1 MB)

```

For more options and information about using `ifconfig`, enter `ifconfig -h` or see the `ifconfig (8)` man page.

13.6.2.4 Configuring Routing with `route`

`route` is a program for manipulating the IP routing table. You can use it to view your routing configuration and to add or remove routes.

REMARQUE : `route` and `ip`

The program `route` is obsolete. Use `ip` instead.

`route` is especially useful if you need quick and comprehensible information about your routing configuration to determine problems with routing. To view your current routing configuration, enter `route -n` as root.

Example 13.12 *Output of the `route -n` Command*

```

route -n
Kernel IP routing table
Destination      Gateway          Genmask          Flags      MSS Window  irtt Iface
10.20.0.0        *                255.255.248.0    U           0 0           0 eth0
link-local       *                255.255.0.0      U           0 0           0 eth0
loopback         *                255.0.0.0        U           0 0           0 lo
default          styx.exam.com    0.0.0.0          UG          0 0           0 eth0

```

For more options and information about using `route`, enter `route -h` or see the `route (8)` man page.

13.6.3 Start-Up Scripts

Apart from the configuration files described above, there are also various scripts that load the network programs while the machine is booting. These are started as soon as the system is switched to one of the *multiuser runlevels*. Some of these scripts are described in Tableau 13.10, « Some Start-Up Scripts for Network Programs » (p. 275).

Tableau 13.10 *Some Start-Up Scripts for Network Programs*

<code>/etc/init.d/network</code>	This script handles the configuration of the network interfaces. If the <code>network</code> service was not started, no network interfaces are implemented.
<code>/etc/init.d/xinetd</code>	Starts <code>xinetd</code> . <code>xinetd</code> can be used to make server services available on the system. For example, it can start <code>vsftpd</code> whenever an FTP connection is initiated.
<code>/etc/init.d/rpcbind</code>	Starts the <code>rpcbind</code> utility that converts RPC program numbers to universal addresses. It is needed for RPC services, such as an NFS server.
<code>/etc/init.d/nfsserver</code>	Starts the NFS server.
<code>/etc/init.d/postfix</code>	Controls the postfix process.
<code>/etc/init.d/ypserv</code>	Starts the NIS server.
<code>/etc/init.d/ypbind</code>	Starts the NIS client.

13.7 smpppd as Dial-up Assistant

Some home users do not have a dedicated line connecting them to the Internet. Instead, they use dial-up connections. Depending on the dial-up method (ISDN or

DSL), the connection is controlled by `ipppd` or `pppd`. Basically, all that needs to be done to go online is to start these programs correctly.

If you have a flat-rate connection that does not generate any additional costs for the dial-up connection, simply start the respective daemon. Control the dial-up connection with a desktop applet or a command-line interface. If the Internet gateway is not the host you are using, you might want to control the dial-up connection by way of a network host.

This is where `smpppd` (SUSE Meta PPP Daemon) is involved. It provides a uniform interface for auxiliary programs and acts in two directions. First, it programs the required `pppd` or `ipppd` and controls its dial-up properties. Second, it makes various providers available to the user programs and transmits information about the current status of the connection. As `smpppd` can also be controlled by way of the network, it is suitable for controlling dial-up connections to the Internet from a workstation in a private subnetwork.

13.7.1 Configuring `smpppd`

The connections provided by `smpppd` are automatically configured by YaST. The actual dial-up programs `KInternet` and `cinternet` are also preconfigured. Manual settings are only required to configure additional features of `smpppd` such as remote control.

The configuration file of `smpppd` is `/etc/smpppd.conf`. By default, it does not enable remote control. The most important options of this configuration file are:

`open-inet-socket = yes/no`

To control `smpppd` via the network, set this option to `yes`. `smpppd` listens on port 3185. If this parameter is set to `yes`, the parameters `bind-address`, `host-range` and `password` must be set accordingly.

`bind-address = ip address`

If a host has several IP addresses, use this parameter to determine at which IP address `smpppd` should accept connections. The default is to listen at all addresses.

`host-range = min ipmax ip`

The parameter `host-range` defines a network range. Hosts whose IP addresses are within this range are granted access to `smpppd`. All hosts not within this range are denied access.

`password = password`

By assigning a password, limit the clients to authorized hosts. As this is a plain-text password, you should not overrate the security it provides. If no password is assigned, all clients are permitted to access smpppd.

`slp-register = yes/no`

With this parameter, the smpppd service can be announced in the network via SLP.

More information about smpppd is available in the `smpppd(8)` and `smpppd.conf(5)` man pages.

13.7.2 Configuring qinternet for Remote Use

qinternet can be used to control a local or remote smpppd. cinternet is the command-line counterpart to the graphical KInternet. To prepare these utilities for use with a remote smpppd, edit the configuration file `/etc/smpppd-c.conf` manually or using qinternet. This file only uses four options:

`sites = list of sites`

list of sites where the front-ends search for smpppd. The front-ends test the options in the order specified here. `local` orders the establishment of a connection to the local smpppd. `gateway` points to an smpppd on the gateway. `config-file` indicates that the connection should be established to the smpppd specified in the `server` and `port` options in `/etc/smpppd-c.conf`. `slp` orders the front-ends to connect to an smpppd found via SLP.

`server = server`

The host on which smpppd runs.

`port = port`

The port on which smpppd runs.

`password = password`

The password selected for smpppd.

SLP Services in the Network

The *service location protocol* (SLP) was developed to simplify the configuration of networked clients within a local network. To configure a network client, including all required services, the administrator traditionally needs detailed knowledge of the servers available in the network. SLP makes the availability of selected services known to all clients in the local network. Applications that support SLP can use the information distributed and be configured automatically.

openSUSE® supports installation using installation sources provided with SLP and contains many system services with integrated support for SLP. YaST and Konqueror both have appropriate front-ends for SLP. You can use SLP to provide networked clients with central functions, such as an installation server, file server, or print server on your system.

IMPORTANT : SLP Support in openSUSE

Services that offer SLP support include cupsd, rsyncd, ypserv, openldap2, ksysguardd, saned, kdm, vnc, login, smpppd, rpasswd, postfix, and sshd (via fish).

14.1 Installation

All packages necessary to use SLP services are installed by default. However, if you want to provide services via SLP, check that the `openslp-server` package is installed. For SLP daemon server configuration install the `yast2-slp-server` package.

14.2 Activating SLP

slpd must run on your system to offer services with SLP. If the machine should only operate as client, and does not offer services, it is not necessary to run slpd. Like most system services in openSUSE, the slpd daemon is controlled by means of a separate `init` script. After the installation, the daemon is inactive by default. To activate it temporarily, run `rcslpd start` as root or `rcslpd stop` to stop it. Perform a restart or status check with `restart` or `status`. If slpd should be always active after booting, enable slpd in YaST *System > System Services (Runlevel)* or run the `insserv slpd` command as root.

14.3 SLP Front-Ends in openSUSE

To find services provided via SLP in your network, use an SLP front-end such as `slptool` (openslp package) or YaST:

slptool

slptool is a command line program that can be used to announce SLP inquiries in the network or announce proprietary services. `slptool --help` lists all available options and functions. For example, to find all time servers that announce themselves in the current network, run the command:

```
slptool findsrvs service:ntp
```

YaST

YaST also provides an SLP browser. However, this browser is not available from the YaST Control Center. To start it, run `yast2 slp` as root user. Click on a *Service Type* on the lefthand side to get more information about a service.

14.4 Installation over SLP

If you have an installation server with openSUSE installation media within your network, this can be registered and offered with SLP. For details, see Section 2.2, « Configuration du serveur contenant les sources de l'installation. » (p. 46). If SLP installation is selected, linuxrc starts an SLP inquiry after the system has booted from the selected boot medium and displays the sources found.

14.5 Providing Services via SLP

Many applications in openSUSE have integrated SLP support through the use of the `libslp` library. If a service has not been compiled with SLP support, use one of the following methods to make it available via SLP:

Static Registration with `/etc/slp.reg.d`

Create a separate registration file for each new service. This is an example for registering a scanner service:

```
## Register a saned service on this system
## en means english language
## 65535 disables the timeout, so the service registration does
## not need refreshes
service:scanner.sane://$HOSTNAME:6566,en,65535
watch-port-tcp=6566
description=SANE scanner daemon
```

The most important line in this file is the *service URL*, which begins with `service:.` This contains the service type (`scanner.sane`) and the address under which the service is available on the server. `$HOSTNAME` is automatically replaced with the full hostname. The name of the TCP port on which the relevant service can be found follows, separated by a colon. Then enter the language in which the service should appear and the duration of registration in seconds. These should be separated from the service URL by commas. Set the value for the duration of registration between 0 and 65535. 0 prevents registration. 65535 removes all restrictions.

The registration file also contains the two variables `watch-port-tcp` and `description`. `watch-port-tcp` links the SLP service announcement to whether the relevant service is active by having `slpd` check the status of the service. The second variable contains a more precise description of the service that is displayed in suitable browsers.

Static Registration with `/etc/slp.reg`

The only difference between this method and the procedure with `/etc/slp.reg.d` is that all services are grouped within a central file.

Dynamic Registration with `slptool`

If a service needs to be registered dynamically without the need of configuration files, use the `slptool` command line utility. The same utility can also be used to deregister an existing service offering without restarting `slpd`.

14.6 For More Information

RFC 2608, 2609, 2610

RFC 2608 generally deals with the definition of SLP. RFC 2609 deals with the syntax of the service URLs used in greater detail and RFC 2610 deals with DHCP via SLP.

<http://www.openslp.org>

The home page of the OpenSLP project.

`/usr/share/doc/packages/openslp`

This directory contains the documentation for SLP coming with the `openslp-server` package, including a `README.SuSE` containing the openSUSE details, the RFCs, and two introductory HTML documents. Programmers who want to use the SLP functions find more information in the *Programmers Guide* that is included in the `openslp-devel` package.

The Domain Name System

DNS (domain name system) is needed to resolve the domain names and hostnames into IP addresses. In this way, the IP address 192.168.2.100 is assigned to the host-name `jupiter`, for example. Before setting up your own name server, read the general information about DNS in Section 13.3, « Name Resolution » (p. 235). The following configuration examples refer to BIND.

15.1 DNS Terminology

Zone

The domain namespace is divided into regions called zones. For instance, if you have `example.com`, you have the `example` section (or zone) of the `com` domain.

DNS server

The DNS server is a server that maintains the name and IP information for a domain. You can have a primary DNS server for master zone, a secondary server for slave zone, or a slave server without any zones for caching.

Master zone DNS server

The master zone includes all hosts from your network and a DNS server master zone stores up-to-date records for all the hosts in your domain.

Slave zone DNS server

A slave zone is a copy of the master zone. The slave zone DNS server obtains its zone data with zone transfer operations from its master server. The slave

zone DNS server responds authoritatively for the zone as long as it has valid (not expired) zone data. If the slave cannot obtain a new copy of the zone data, it stops responding for the zone.

Forwarder

Forwarders are DNS servers to which your DNS server should send queries it cannot answer. To enable different configuration sources in one configuration, `netconfig` is used (see also `man 8 netconfig`).

Record

The record is information about name and IP address. Supported records and their syntax are described in BIND documentation. Some special records are:

NS record

An NS record tells name servers which machines are in charge of a given domain zone.

MX record

The MX (mail exchange) records describe the machines to contact for directing mail across the Internet.

SOA record

SOA (Start of Authority) record is the first record in a zone file. The SOA record is used when using DNS to synchronize data between multiple computers.

15.2 Installation

To install a DNS server, start YaST and select *Software > Software Management*. Choose *View > Patterns* and select *DHCP and DNS Server*. Confirm the installation of the dependent packages to finish the installation process.

15.3 Configuration with YaST

Use the YaST DNS module to configure a DNS server for the local network. When starting the module for the first time, a wizard starts, prompting you to make a few decisions concerning administration of the server. Completing this initial setup produces a basic server configuration. Use the expert mode to deal with more

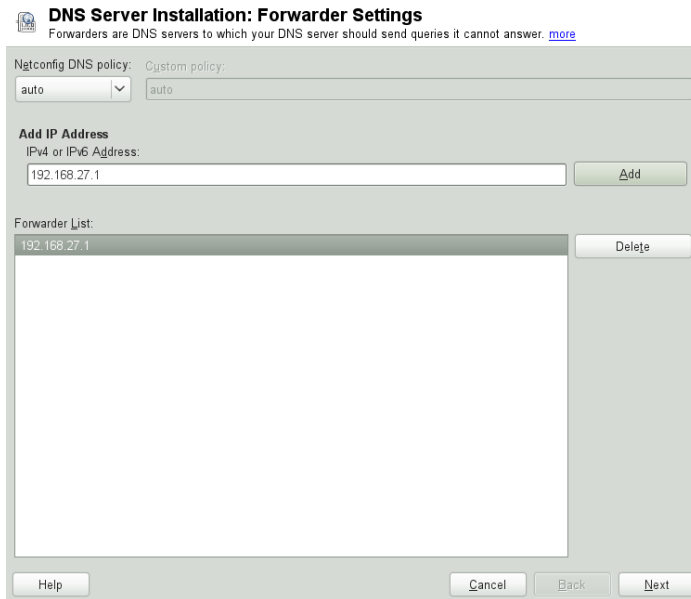
advanced configuration tasks, such as setting up ACLs, logging, TSIG keys, and other options.

15.3.1 Wizard Configuration

The wizard consists of three steps or dialogs. At the appropriate places in the dialogs, you are given the opportunity to enter the expert configuration mode.

- 1 When starting the module for the first time, the *Forwarder Settings* dialog, shown in Figure 15.1, « DNS Server Installation: Forwarder Settings » (p. 285), opens. The *Netconfig DNS Policy* decides which devices should provide forwarders or whether you want to supply your own *Forwarder List*. For more information about *netconfig*, see `man 8 netconfig`.

Figure 15.1 *DNS Server Installation: Forwarder Settings*

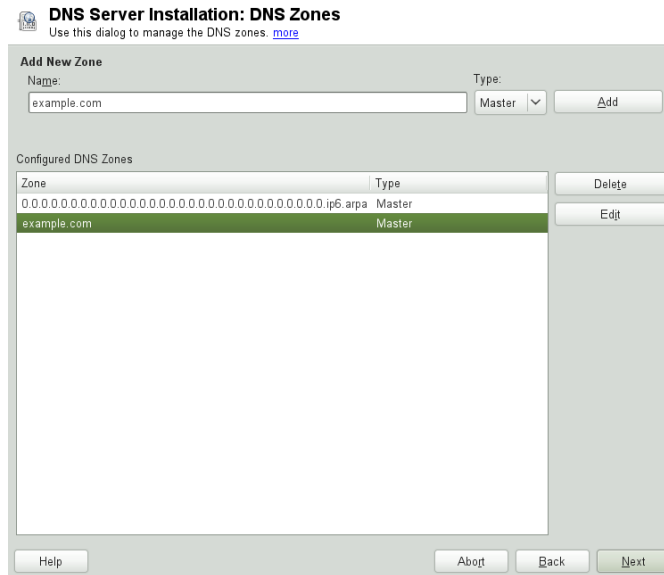


Forwarders are DNS servers to which your DNS server sends queries it cannot answer itself. Enter their IP address and click *Add*.

- 2 The *DNS Zones* dialog consists of several parts and is responsible for the management of zone files, described in Section 15.6, « Zone Files » (p. 300). For a

new zone, provide a name for it in *Name*. To add a reverse zone, the name must end in `.in-addr.arpa`. Finally, select the *Type* (master, slave, or forward). See Figure 15.2, « DNS Server Installation: DNS Zones » (p. 286). Click *Edit* to configure other settings of an existing zone. To remove a zone, click *Delete*.

Figure 15.2 *DNS Server Installation: DNS Zones*



- 3** In the final dialog, you can open the DNS port in the firewall by clicking *Open Port in Firewall*. Then decide whether to start the DNS server when booting (*On* or *Off*). You can also activate LDAP support. See Figure 15.3, « DNS Server Installation: Finish Wizard » (p. 287).

DNS Server Installation: Finish Wizard

Check the entered settings before finishing the configuration. [more](#)

☐ Open Port in Firewall Firewall Details...

Firewall port is closed

☒ LDAP Support Active

Start-Up Behavior

☐ On: Start Now and When Booting

☒ Off: Only Start Manually

- Forwarders: 192.168.27.1
- Domains: ., localhost, 0.0.127.in-addr.arpa, 0.ip6.arpa, example.com

DNS Server Expert Configuration...

After starting the module, YaST opens a window displaying several configuration options. Completing it results in a DNS server configuration with the basic functions in place:

Under *Start-Up*, define whether the DNS server should be started when the booting the system or manually. To start the DNS server immediately, click *Start DNS Server Now*. To stop the DNS server, click *Stop DNS Server Now*. To save the current settings, select *Save Settings and Reload DNS Server Now*. You can open the DNS port in the firewall with *Open Port in Firewall* and modify the firewall settings with *Firewall Details*.

The Domain Name System 287

15.3.2.2 Forwarders

If your local DNS server cannot answer a request, it tries to forward the request to a *Forwarder*, if configured so. This forwarder may be added manually to the *Forwarder List*. If the forwarder is not static like in dial-up connections, *netconfig* handles the configuration. For more information about *netconfig*, see `man 8 netconfig`.

15.3.2.3 Basic Options

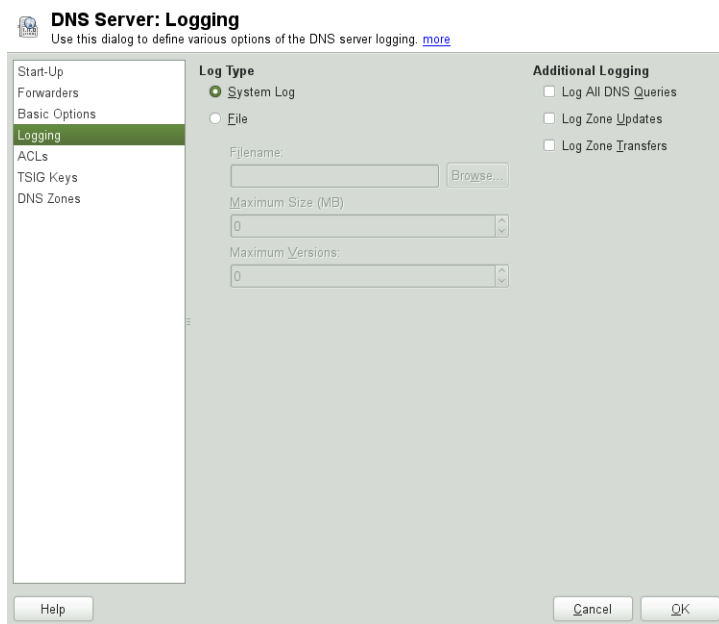
In this section, set basic server options. From the *Option* menu, select the desired item then specify the value in the corresponding entry field. Include the new entry by selecting *Add*.

15.3.2.4 Logging

To set what the DNS server should log and how, select *Logging*. Under *Log Type*, specify where the DNS server should write the log data. Use the systemwide log file `/var/log/messages` by selecting *System Log* or specify a different file by selecting *File*. In the latter case, additionally specify a name, the maximum file size in megabytes and the number of logfile versions to store.

Further options are available under *Additional Logging*. Enabling *Log All DNS Queries* causes *every* query to be logged, in which case the log file could grow extremely large. For this reason, it is not a good idea to enable this option for other than debugging purposes. To log the data traffic during zone updates between DHCP and DNS server, enable *Log Zone Updates*. To log the data traffic during a zone transfer from master to slave, enable *Log Zone Transfer*. See Figure 15.4, « DNS Server: Logging » (p. 289).

Figure 15.4 *DNS Server: Logging*



15.3.2.5 ACLs

Use this dialog to define ACLs (access control lists) to enforce access restrictions. After providing a distinct name under *Name*, specify an IP address (with or without netmask) under *Value* in the following fashion:

```
{ 192.168.1/24; }
```

The syntax of the configuration file requires that the address ends with a semicolon and is put into curly braces.

15.3.2.6 TSIG Keys

The main purpose of TSIGs (transaction signatures) is to secure communications between DHCP and DNS servers. They are described in Section 15.8, « Secure Transactions » (p. 304).

To generate a TSIG key, enter a distinctive name in the field labeled *Key ID* and specify the file where the key should be stored (*Filename*). Confirm your choices with *Generate*.

To use a previously created key, leave the *Key ID* field blank and select the file where it is stored under *Filename*. After that, confirm with *Add*.

15.3.2.7 DNS Zones (Adding a Slave Zone)

To add a slave zone, select *DNS Zones*, choose the zone type *Slave*, write the name of the new zone, and click *Add*.

In the *Zone Editor* sub-dialog under *Master DNS Server IP*, specify the master from which the slave should pull its data. To limit access to the server, select one of the ACLs from the list.

15.3.2.8 DNS Zones (Adding a Master Zone)

To add a master zone, select *DNS Zones*, choose the zone type *Master*, write the name of the new zone, and click *Add*. When adding a master zone, a reverse zone is also needed. For example, when adding the zone `example.com` that points to hosts in a subnet `192.168.1.0/24`, you should also add a reverse zone for the IP-address range covered. By definition, this should be named `1.168.192.in-addr.arpa`.

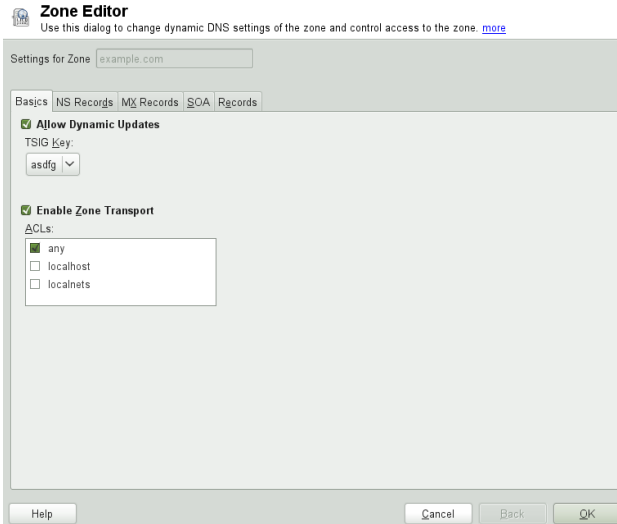
15.3.2.9 DNS Zones (Editing a Master Zone)

To edit a master zone, select *DNS Zones*, select the master zone from the table, and click *Edit*. The dialog consists of several pages: *Basics* (the one opened first), *NS Records*, *MX Records*, *SOA*, and *Records*.

The basic dialog, shown in Figure 15.5, « DNS Server: Zone Editor (Basics) » (p. 291), lets you define settings for dynamic DNS and access options for zone transfers to clients and slave name servers. To permit the dynamic updating of zones, select *Allow Dynamic Updates* as well as the corresponding TSIG key. The key must have been defined before the update action starts. To enable zone transfers, select the corresponding ACLs. ACLs must have been defined already.

In the *Basics* dialog, select whether to enable zone transfers. Use the listed ACLs to define who can download zones.

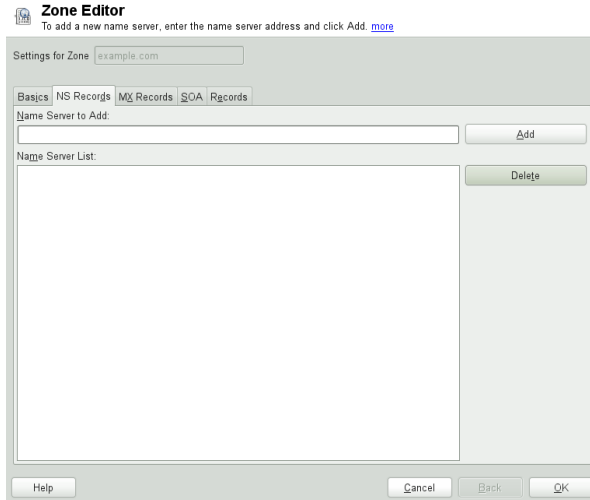
Figure 15.5 *DNS Server: Zone Editor (Basics)*



Zone Editor (NS Records)

The *NS Records* dialog allows you to define alternative name servers for the zones specified. Make sure that your own name server is included in the list. To add a record, enter its name under *Name Server to Add* then confirm with *Add*. See Figure 15.6, « DNS Server: Zone Editor (NS Records) » (p. 292).

Figure 15.6 *DNS Server: Zone Editor (NS Records)*



Zone Editor (MX Records)

To add a mail server for the current zone to the existing list, enter the corresponding address and priority value. After doing so, confirm by selecting *Add*. See Figure 15.7, « DNS Server: Zone Editor (MX Records) » (p. 293).

Figure 15.7 *DNS Server: Zone Editor (MX Records)*

The screenshot shows the 'Zone Editor' window for 'example.com'. The 'MX Records' tab is selected. The 'Mail Server to Add' section has an 'Address' field and a 'Priority' dropdown set to '0', with an 'Add' button. Below is a 'Mail Relay List' table with columns 'Mail Server' and 'Priority', and a 'Delete' button. The table is empty, showing 'No entries.' At the bottom are 'Help', 'Cancel', 'Back', and 'OK' buttons.

Zone Editor
To add a new mail server, enter the Address and Priority and click Add. [more](#)

Settings for Zone:

Basics | NS Records | **MX Records** | SOA | Records

Mail Server to Add

Address:

Priority:

Add

Mail Relay List

Mail Server	Priority
No entries.	

Delete

Help Cancel Back OK

Zone Editor (SOA)

This page allows you to create SOA (start of authority) records. For an explanation of the individual options, refer to Exemple 15.6, « The `/var/lib/named/example.com.zone` File » (p. 300).

Figure 15.8 DNS Server: Zone Editor (SOA)

Zone Editor
Set the entries of the SOA record. [more](#)

Settings for Zone:

Basics | NS Records | MX Records | **SOA** | Records

Serial:

TTL: Unit:

Refresh: Unit:

Retry: Unit:

Expiration: Unit:

Minimum: Unit:

Zone Editor (Records)

This dialog manages name resolution. In *Record Key*, enter the hostname then select its type. *A-Record* represents the main entry. The value for this should be an IP address. *CNAME* is an alias. Use the types *NS* and *MX* for detailed or partial records that expand on the information provided in the *NS Records* and *MX Records* tabs. These three types resolve to an existing *A* record. *PTR* is for reverse zones. It is the opposite of an *A* record, for example:

```
hostname.example.com. IN A 192.168.0.1
1.0.168.192.in-addr.arpa IN PTR hostname.example.com.
```

REMARQUE : Editing the Reverse Zone

After adding a forward zone, go back to the main menu and select the reverse zone for editing. There in the tab *Basics* activate the checkbox *Automatically Generate Records From* and select your forward zone. That way, all changes to the forward zone are automatically updated in the reverse zone.

15.4 Starting the BIND Name Server

On a openSUSE® system, the name server BIND (*Berkeley Internet Name Domain*) comes preconfigured so it can be started right after installation without any problems. If you already have a functioning Internet connection and have entered 127.0.0.1 as the name server address for localhost in `/etc/resolv.conf`, you normally already have a working name resolution without needing to know the DNS of the provider. BIND carries out name resolution via the root name server, a notably slower process. Normally, the DNS of the provider should be entered with its IP address in the configuration file `/etc/named.conf` under `forwarders` to ensure effective and secure name resolution. If this works so far, the name server runs as a pure *caching-only* name server. Only when you configure its own zones it becomes a proper DNS. Find a simple example documented in `/usr/share/doc/packages/bind/config`.

ASTUCE : Automatic Adaptation of the Name Server Information

Depending on the type of Internet connection or the network connection, the name server information can automatically be adapted to the current conditions. To do this, set the `NETCONFIG_DNS_POLICY` variable in the `/etc/sysconfig/network/config` file to `auto`.

However, do not set up an official domain until one is assigned to you by the responsible institution. Even if you have your own domain and it is managed by the provider, you are better off not using it, because BIND would otherwise not forward requests for this domain. The Web server at the provider, for example, would not be accessible for this domain.

To start the name server, enter the command `rndc start` as root. If « done » appears to the right in green then named (as the name server process is called) has been started successfully. Test the name server immediately on the local system with the `host` or `dig` programs, which should return `localhost` as the default server with the address 127.0.0.1. If this is not the case, `/etc/resolv.conf` probably contains an incorrect name server entry or the file does not exist at all. For the first test, enter `host 127.0.0.1`, which should always work. If you get an error message, use `rndc status` to see whether the server is actually running. If the name server does not start or behaves unexpectedly, you can usually find the cause in the log file `/var/log/messages`.

To use the name server of the provider (or one already running on your network) as the forwarder, enter the corresponding IP address or addresses in the `options` section under `forwarders`. The addresses included in Exemple 15.1, « Forwarding

Options in `named.conf` » (p. 296) are just examples. Adjust these entries to your own setup.

Example 15.1 *Forwarding Options in `named.conf`*

```
options {
    directory "/var/lib/named";
    forwarders { 10.11.12.13; 10.11.12.14; };
    listen-on { 127.0.0.1; 192.168.1.116; };
    allow-query { 127/8; 192.168/16 };
    notify no;
};
```

The `options` entry is followed by entries for the zone, `localhost`, and `0.0.127.in-addr.arpa`. The `type hint` entry under `«.»` should always be present. The corresponding files do not need to be modified and should work as they are. Also make sure that each entry is closed with a `«;»` and that the curly braces are in the correct places. After changing the configuration file `/etc/named.conf` or the zone files, tell BIND to reread them with `rndc reload`. Achieve the same by stopping and restarting the name server with `rndc restart`. Stop the server at any time by entering `rndc stop`.

15.5 The `/etc/named.conf` Configuration File

All the settings for the BIND name server itself are stored in the `/etc/named.conf` file. However, the zone data for the domains to handle (consisting of the hostnames, IP addresses, and so on) are stored in separate files in the `/var/lib/named` directory. The details of this are described later.

`/etc/named.conf` is roughly divided into two areas. One is the `options` section for general settings and the other consists of `zone` entries for the individual domains. A logging section and `acl` (access control list) entries are optional. Comment lines begin with a `#` sign or `//`. A minimal `/etc/named.conf` is shown in Example 15.2, « A Basic `/etc/named.conf` » (p. 296).

Example 15.2 *A Basic `/etc/named.conf`*

```
options {
    directory "/var/lib/named";
    forwarders { 10.0.0.1; };
    notify no;
```

```

};

zone "localhost" in {
    type master;
    file "localhost.zone";
};

zone "0.0.127.in-addr.arpa" in {
    type master;
    file "127.0.0.zone";
};

zone "." in {
    type hint;
    file "root.hint";
};

```

15.5.1 Important Configuration Options

`directory "filename";`

Specifies the directory in which BIND can find the files containing the zone data. Usually, this is `/var/lib/named`.

`forwarders { ip-address; };`

Specifies the name servers (mostly of the provider) to which DNS requests should be forwarded if they cannot be resolved directly. Replace *ip-address* with an IP address like `192.168.1.116`.

`forward first;`

Causes DNS requests to be forwarded before an attempt is made to resolve them via the root name servers. Instead of `forward first`, `forward only` can be written to have all requests forwarded and none sent to the root name servers. This makes sense for firewall configurations.

`listen-on port 53 { 127.0.0.1; ip-address; };`

Tells BIND on which network interfaces and port to accept client queries. `port 53` does not need to be specified explicitly, because 53 is the default port. Enter `127.0.0.1` to permit requests from the local host. If you omit this entry entirely, all interfaces are used by default.

`listen-on-v6 port 53 {any; };`

Tells BIND on which port it should listen for IPv6 client requests. The only alternative to `any` is `none`. As far as IPv6 is concerned, the server only accepts wild card addresses.

`query-source address * port 53;`

This entry is necessary if a firewall is blocking outgoing DNS requests. This tells BIND to post requests externally from port 53 and not from any of the high ports above 1024.

`query-source-v6 address * port 53;`

Tells BIND which port to use for IPv6 queries.

`allow-query { 127.0.0.1; net; };`

Defines the networks from which clients can post DNS requests. Replace *net* with address information like `192.168.2.0/24`. The `/24` at the end is an abbreviated expression for the netmask (in this case `255.255.255.0`).

`allow-transfer ! *;;`

Controls which hosts can request zone transfers. In the example, such requests are completely denied with `! *`. Without this entry, zone transfers can be requested from anywhere without restrictions.

`statistics-interval 0;`

In the absence of this entry, BIND generates several lines of statistical information per hour in `/var/log/messages`. Set it to 0 to suppress these statistics completely or set an interval in minutes.

`cleaning-interval 720;`

This option defines at which time intervals BIND clears its cache. This triggers an entry in `/var/log/messages` each time it occurs. The time specification is in minutes. The default is 60 minutes.

`interface-interval 0;`

BIND regularly searches the network interfaces for new or nonexistent interfaces. If this value is set to 0, this is not done and BIND only listens at the interfaces detected at start-up. Otherwise, the interval can be defined in minutes. The default is sixty minutes.

`notify no;`

`no` prevents other name servers from being informed when changes are made to the zone data or when the name server is restarted.

For a list of available options, read the manual page `man 5 named.conf`.

15.5.2 Logging

What, how, and where logging takes place can be extensively configured in BIND. Normally, the default settings should be sufficient. Example 15.3, « Entry to Disable Logging » (p. 299), shows the simplest form of such an entry and completely suppresses any logging.

Example 15.3 *Entry to Disable Logging*

```
logging {  
    category default { null; };  
};
```

15.5.3 Zone Entries

Example 15.4 *Zone Entry for example.com*

```
zone "example.com" in {  
    type master;  
    file "example.com.zone";  
    notify no;  
};
```

After `zone`, specify the name of the domain to administer (`example.com`) followed by `in` and a block of relevant options enclosed in curly braces, as shown in Example 15.4, « Zone Entry for example.com » (p. 299). To define a *slave zone*, switch the `type` to `slave` and specify a name server that administers this zone as `master` (which, in turn, may be a slave of another master), as shown in Example 15.5, « Zone Entry for example.net » (p. 299).

Example 15.5 *Zone Entry for example.net*

```
zone "example.net" in {  
    type slave;  
    file "slave/example.net.zone";  
    masters { 10.0.0.1; };  
};
```

The zone options:

`type master;`

By specifying `master`, tell BIND that the zone is handled by the local name server. This assumes that a zone file has been created in the correct format.

`type slave;`

This zone is transferred from another name server. It must be used together with `masters`.

type hint;

The zone `.` of the `hint` type is used to set the root name servers. This zone definition can be left as is.

file `example.com.zone` or file « `slave/example.net.zone` »;

This entry specifies the file where zone data for the domain is located. This file is not required for a slave, because this data is pulled from another name server. To differentiate master and slave files, use the directory `slave` for the slave files.

masters { `server-ip-address`; };

This entry is only needed for slave zones. It specifies from which name server the zone file should be transferred.

allow-update { `! *`; };

This option controls external write access, which would allow clients to make a DNS entry—something not normally desirable for security reasons. Without this entry, zone updates are not allowed at all. The above entry achieves the same because `! *` effectively bans any such activity.

15.6 Zone Files

Two types of zone files are needed. One assigns IP addresses to hostnames and the other does the reverse: it supplies a hostname for an IP address.

ASTUCE : Using the Dot (Period, Fullstop) in Zone Files

The `" . "` has an important meaning in the zone files. If hostnames are given without a final `.`, the zone is appended. Complete hostnames specified with a full domain name must end with a `.` to avoid having the domain added to it again. A missing or wrongly placed `" . "` is probably the most frequent cause of name server configuration errors.

The first case to consider is the zone file `example.com.zone`, responsible for the domain `example.com`, shown in Exemple 15.6, « The `/var/lib/named/example.com.zone` File » (p. 300).

Exemple 15.6 *The `/var/lib/named/example.com.zone` File*

```
1. $TTL 2D
2. example.com. IN SOA      dns root.example.com. (
3.                  2003072441 ; serial
4.                  1D        ; refresh
```

```

5.                2H                ; retry
6.                1W                ; expiry
7.                2D )              ; minimum
8.
9.                IN NS             dns
10.               IN MX             10 mail
11.
12. gate          IN A              192.168.5.1
13.               IN A              10.0.0.1
14. dns           IN A              192.168.1.116
15. mail          IN A              192.168.3.108
16. jupiter       IN A              192.168.2.100
17. venus         IN A              192.168.2.101
18. saturn        IN A              192.168.2.102
19. mercury       IN A              192.168.2.103
20. ntp           IN CNAME          dns
21. dns6          IN A6 0           2002:c0a8:174::

```

Line 1:

\$TTL defines the default time to live that should apply to all the entries in this file. In this example, entries are valid for a period of two days (2 D).

Line 2:

This is where the SOA (start of authority) control record begins:

- The name of the domain to administer is `example.com` in the first position. This ends with `"."`, because otherwise the zone would be appended a second time. Alternatively, `@` can be entered here, in which case the zone would be extracted from the corresponding entry in `/etc/named.conf`.
- After `IN SOA` is the name of the name server in charge as master for this zone. The name is expanded from `dns` to `dns.example.com`, because it does not end with a `"."`.
- An e-mail address of the person in charge of this name server follows. Because the `@` sign already has a special meaning, `"."` is entered here instead. For `root@example.com` the entry must read `root.example.com..` The `"."` must be included at the end to prevent the zone from being added.
- The `(` includes all lines up to `)` into the SOA record.

Line 3:

The `serial` number is an arbitrary number that is increased each time this file is changed. It is needed to inform the secondary name servers (slave servers)

of changes. For this, a 10 digit number of the date and run number, written as YYYYMMDDNN, has become the customary format.

Line 4:

The `refresh rate` specifies the time interval at which the secondary name servers verify the `zone serial number`. In this case, one day.

Line 5:

The `retry rate` specifies the time interval at which a secondary name server, in case of error, attempts to contact the primary server again. Here, two hours.

Line 6:

The `expiration time` specifies the time frame after which a secondary name server discards the cached data if it has not regained contact to the primary server. Here, a week.

Line 7:

The last entry in the SOA record specifies the `negative caching TTL`—the time for which results of unresolved DNS queries from other servers may be cached.

Line 9:

The `IN NS` specifies the name server responsible for this domain. `dns` is extended to `dns.example.com` because it does not end with a `" . "`. There can be several lines like this—one for the primary and one for each secondary name server. If `notify` is not set to `no` in `/etc/named.conf`, all the name servers listed here are informed of the changes made to the zone data.

Line 10:

The `MX` record specifies the mail server that accepts, processes, and forwards e-mails for the domain `example.com`. In this example, this is the host `mail.example.com`. The number in front of the hostname is the preference value. If there are multiple `MX` entries, the mail server with the smallest value is taken first and, if mail delivery to this server fails, an attempt is made with the next higher value.

Lines 12–19:

These are the actual address records where one or more IP addresses are assigned to hostnames. The names are listed here without a `" . "` because they do not include their domain, so `example.com` is added to all of them. Two IP addresses are assigned to the host `gate`, as it has two network cards. Wherever

the host address is a traditional one (IPv4), the record is marked with A. If the address is an IPv6 address, the entry is marked with AAAA.

REMARQUE : IPv6 Syntax

The IPv6 record has a slightly different syntax than IPv4. Because of the fragmentation possibility, it is necessary to provide information about missed bits before the address. To just fill up the IPv6 address with the needed number of « 0 », add two colons at the correct place in the address.

```
pluto      AAAA 2345:00C1:CA11::1234:5678:9ABC:DEF0
pluto      AAAA 2345:00D2:DA11::1234:5678:9ABC:DEF0
```

Line 20:
The alias `ntp` can be used to address `dns` (CNAME means *canonical name*).

The pseudodomain `in-addr.arpa` is used for the reverse lookup of IP addresses into hostnames. It is appended to the network part of the address in reverse notation. So `192.168` is resolved into `168.192.in-addr.arpa`. See Exemple 15.7, « Reverse Lookup » (p. 303).

Exemple 15.7 Reverse Lookup

```
1. $TTL 2D
2. 168.192.in-addr.arpa.    IN SOA dns.example.com. root.example.com. (
3.                          2003072441      ; serial
4.                          1D              ; refresh
5.                          2H              ; retry
6.                          1W              ; expiry
7.                          2D )            ; minimum
8.
9.                          IN NS          dns.example.com.
10.
11. 1.5                     IN PTR         gate.example.com.
12. 100.3                   IN PTR         www.example.com.
13. 253.2                   IN PTR         cups.example.com.
```

Line 1:
\$TTL defines the standard TTL that applies to all entries here.

Line 2:
The configuration file should activate reverse lookup for the network `192.168`. Given that the zone is called `168.192.in-addr.arpa`, it should not be added to the hostnames. Therefore, all hostnames are entered in their complete

form—with their domain and with a " ." at the end. The remaining entries correspond to those described for the previous `example.com` example.

Lines 3–7:

See the previous example for `example.com`.

Line 9:

Again this line specifies the name server responsible for this zone. This time, however, the name is entered in its complete form with the domain and a " ." at the end.

Lines 11–13:

These are the pointer records hinting at the IP addresses on the respective hosts. Only the last part of the IP address is entered at the beginning of the line, without the " ." at the end. Appending the zone to this (without the `.in-addr.arpa`) results in the complete IP address in reverse order.

Normally, zone transfers between different versions of BIND should be possible without any problems.

15.7 Dynamic Update of Zone Data

The term *dynamic update* refers to operations by which entries in the zone files of a master server are added, changed, or deleted. This mechanism is described in RFC 2136. Dynamic update is configured individually for each zone entry by adding an optional `allow-update` or `update-policy` rule. Zones to update dynamically should not be edited by hand.

Transmit the entries to update to the server with the command `nsupdate`. For the exact syntax of this command, check the manual page for `nsupdate` (`man 8 nsupdate`). For security reasons, any such update should be performed using TSIG keys as described in Section 15.8, « Secure Transactions » (p. 304).

15.8 Secure Transactions

Secure transactions can be made with the help of transaction signatures (TSIGs) based on shared secret keys (also called TSIG keys). This section describes how to generate and use such keys.

Secure transactions are needed for communication between different servers and for the dynamic update of zone data. Making the access control dependent on keys is much more secure than merely relying on IP addresses.

Generate a TSIG key with the following command (for details, see `man dnssec-keygen`):

```
dnssec-keygen -a hmac-md5 -b 128 -n HOST host1-host2
```

This creates two files with names similar to these:

```
Khost1-host2.+157+34265.private Khost1-host2.+157+34265.key
```

The key itself (a string like `ejIkuCyyGJwwuN3xAteKgg==`) is found in both files. To use it for transactions, the second file (`Khost1-host2.+157+34265.key`) must be transferred to the remote host, preferably in a secure way (using `scp`, for example). On the remote server, the key must be included in the `/etc/named.conf` file to enable a secure communication between `host1` and `host2`:

```
key host1-host2 {  
    algorithm hmac-md5;  
    secret "ejIkuCyyGJwwuN3xAteKgg=";  
};
```

AVERTISSEMENT : File Permissions of `/etc/named.conf`

Make sure that the permissions of `/etc/named.conf` are properly restricted. The default for this file is `0640`, with the owner being `root` and the group `named`. As an alternative, move the keys to an extra file with specially limited permissions, which is then included from `/etc/named.conf`. To include an external file, use:

```
include "filename"
```

Replace `filename` with an absolute path to your file with keys.

To enable the server `host1` to use the key for `host2` (which has the address `10.1.2.3` in this example), the server's `/etc/named.conf` must include the following rule:

```
server 10.1.2.3 {  
    keys { host1-host2. ; };  
};
```

Analogous entries must be included in the configuration files of `host2`.

Add TSIG keys for any ACLs (access control lists, not to be confused with file system ACLs) that are defined for IP addresses and address ranges to enable transaction security. The corresponding entry could look like this:

```
allow-update { key host1-host2. ;};
```

This topic is discussed in more detail in the *BIND Administrator Reference Manual* under `update-policy`.

15.9 DNS Security

DNSSEC, or DNS security, is described in RFC 2535. The tools available for DNSSEC are discussed in the BIND Manual.

A zone considered secure must have one or several zone keys associated with it. These are generated with `dnssec-keygen`, just like the host keys. The DSA encryption algorithm is currently used to generate these keys. The public keys generated should be included in the corresponding zone file with an `$INCLUDE` rule.

With the command `dnssec-signzone`, you can create sets of generated keys (`keyset-files`), transfer them to the parent zone in a secure manner, and sign them. This generates the files to include for each zone in `/etc/named.conf`.

15.10 For More Information

For additional information, refer to the *BIND Administrator Reference Manual* from package `bind-doc`, which is installed under `/usr/share/doc/packages/bind/`. Consider additionally consulting the RFCs referenced by the manual and the manual pages included with BIND. `/usr/share/doc/packages/bind/README.SuSE` contains up-to-date information about BIND in openSUSE.

DHCP

The purpose of the *Dynamic Host Configuration Protocol* (DHCP) is to assign network settings centrally (from a server) rather than configuring them locally on each and every workstation. A host configured to use DHCP does not have control over its own static address. It is enabled to configure itself completely and automatically according to directions from the server. If you use the NetworkManager on the client side, you do not need to configure the client at all. This is useful if you have changing environments and only one interface active at a time. Never use NetworkManager on a machine that runs a DHCP server.

One way to configure a DHCP server is to identify each client using the hardware address of its network card (which should be fixed in most cases), then supply that client with identical settings each time it connects to the server. DHCP can also be configured to assign addresses to each relevant client dynamically from an address pool set up for this purpose. In the latter case, the DHCP server tries to assign the same address to the client each time it receives a request, even over extended periods. This works only if the network does not have more clients than addresses.

DHCP makes life easier for system administrators. Any changes, even bigger ones, related to addresses and the network configuration in general can be implemented centrally by editing the server's configuration file. This is much more convenient than reconfiguring numerous workstations. It is also much easier to integrate machines, particularly new machines, into the network, because they can be given an IP address from the pool. Retrieving the appropriate network settings from a DHCP server is especially useful in case of laptops regularly used in different networks.

In this chapter, the DHCP server will run in the same subnet as the workstations, 192.168.2.0/24 with 192.168.2.1 as gateway. It has the fixed IP address 192.168.2.254 and serves two address ranges, 192.168.2.10 to 192.168.2.20 and 192.168.2.100 to 192.168.2.200.

A DHCP server supplies not only the IP address and the netmask, but also the host-name, domain name, gateway, and name server addresses for the client to use. In addition to that, DHCP allows a number of other parameters to be configured in a centralized way, for example, a time server from which clients may poll the current time or even a print server.

16.1 Configuring a DHCP Server with YaST

To install a DHCP server, start YaST and select *Software > Software Management*. Choose *Filter > Patterns* and select *DHCP and DNS Server*. Confirm the installation of the dependent packages to finish the installation process.

IMPORTANT : LDAP Support

The YaST DHCP module can be set up to store the server configuration locally (on the host that runs the DHCP server) or to have its configuration data managed by an LDAP server. If you want to use LDAP, set up your LDAP environment before configuring the DHCP server.

For more information about LDAP, see Chapitre 4, *LDAP—A Directory Service* (↑Security Guide).

The YaST DHCP module (`yast2-dhcp-server`) allows you to set up your own DHCP server for the local network. The module can run in wizard mode or expert configuration mode.

16.1.1 Initial Configuration (Wizard)

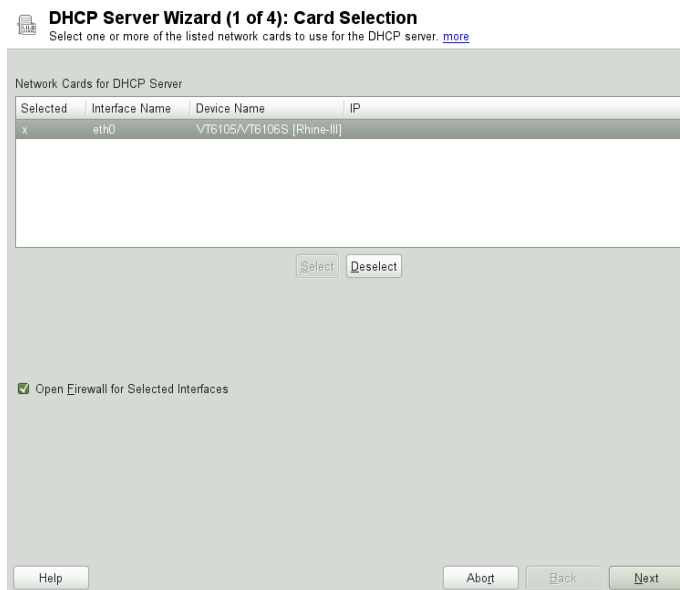
When the module is started for the first time, a wizard starts, prompting you to make a few basic decisions concerning server administration. Completing this initial setup

produces a very basic server configuration that should function in its essential aspects. The expert mode can be used to deal with more advanced configuration tasks.

Card Selection

In the first step, YaST looks for the network interfaces available on your system and displays them in a list. From the list, select the interface to which the DHCP server should listen and click *Select*. After this, select *Open Firewall for Selected Interfaces* to open the firewall for this interface, and click *Next*. See Figure 16.1, « DHCP Server: Card Selection » (p. 309).


Figure 16.1 *DHCP Server: Card Selection*



Global Settings

Use the check box to determine whether your DHCP settings should be automatically stored by an LDAP server. In the entry fields, provide the network specifics for all clients the DHCP server should manage. These specifics are the domain name, address of a time server, addresses of the primary and secondary name server, addresses of a print and a WINS server (for a mixed network with both Windows and Linux clients), gateway address, and lease time. See Figure 16.2, « DHCP Server: Global Settings » (p. 310).

Figure 16.2 *DHCP Server: Global Settings*

 **DHCP Server Wizard (2 of 4): Global Settings**
To store the DHCP configuration in LDAP, enable LDAP Support. [more](#)

☐ LDAP Support

DHCP Server Name (optional)

Domain Name:

Primary Name Server IP:

Secondary Name Server IP:

Default Gateway (Router)

NTP Time Server:

Print Server:

WINS Server:


Default Lease Time:

Units:

Dynamic DHCP

In this step, configure how dynamic IP addresses should be assigned to clients. To do so, specify an IP range from which the server can assign addresses to DHCP clients. All these addresses must be covered by the same netmask. Also specify the lease time during which a client may keep its IP address without needing to request an extension of the lease. Optionally, specify the maximum lease time—the period during which the server reserves an IP address for a particular client. See Figure 16.3, « DHCP Server: Dynamic DHCP » (p. 311).

Figure 16.3 *DHCP Server: Dynamic DHCP*

 **DHCP Server Wizard (3 of 4): Dynamic DHCP**
Here you can view the information about the current subnet, such as its address, netmask and minimum and maximum IP addresses available for the clients. [more](#)

Subnet Information	
Current Network:	Current Netmask:
192.168.2.0	255.255.255.0
Netmask Bits:	
24	
Minimum IP Address:	Maximum IP Address:
192.168.2.1	192.168.2.254

IP Address Range	
First IP Address:	Last IP Address:
192.168.2.100	192.168.2.139

☐ Allow Dynamic BOOTP

Lease Time	
Default:	Units:
4	Hours
Maximum:	Units:
2	Days

[Synchronize DNS Server...](#)

[Help](#) [About](#) [Back](#) [Next](#)

Finishing the Configuration and Setting the Start Mode

After the third part of the configuration wizard, a last dialog is shown in which you can define how the DHCP server should be started. Here, specify whether to start the DHCP server automatically when the system is booted or manually when needed (for example, for testing purposes). Click *Finish* to complete the configuration of the server. See Figure 16.4, « DHCP Server: Start-Up » (p. 311).

Figure 16.4 *DHCP Server: Start-Up*

 **DHCP Server Wizard (4 of 4): Start-Up**
To start the service every time your computer is booted, set When Booting. [more](#)

Service Start

☐ When Booting

☒ Manually

[DHCP Server Expert Configuration...](#)

[Help](#) [About](#) [Back](#) [Finish](#)

16.2 DHCP Software Packages

Both the DHCP server and the DHCP clients are available for openSUSE. The DHCP server available is `dhcpcd` (published by the Internet Systems Consortium). On the client side, choose between two different DHCP client programs: `dhcp-client` (also from ISC) and the DHCP client daemon in the `dhcpcd` package.

openSUSE installs `dhcpcd` by default. The program is very easy to handle and is launched automatically on each system boot to watch for a DHCP server. It does not need a configuration file to do its job and works out of the box in most standard setups. For more complex situations, use the ISC `dhcp-client`, which is controlled by means of the configuration file `/etc/dhclient.conf`.

16.3 The DHCP Server `dhcpcd`

The core of any DHCP system is the dynamic host configuration protocol daemon. This server *leases* addresses and watches how they are used, according to the settings defined in the configuration file `/etc/dhcpd.conf`. By changing the parameters and values in this file, a system administrator can influence the program's behavior in numerous ways. Look at the basic sample `/etc/dhcpd.conf` file in Example 16.1, « The Configuration File `/etc/dhcpd.conf` » (p. 312).

Example 16.1 *The Configuration File `/etc/dhcpd.conf`*

```
default-lease-time 600;           # 10 minutes
max-lease-time 7200;             # 2  hours

option domain-name "example.com";
option domain-name-servers 192.168.1.116;
option broadcast-address 192.168.2.255;
option routers 192.168.2.1;
option subnet-mask 255.255.255.0;

subnet 192.168.2.0 netmask 255.255.255.0
{
    range 192.168.2.10 192.168.2.20;
    range 192.168.2.100 192.168.2.200;
}
```

This simple configuration file should be sufficient to get the DHCP server to assign IP addresses in the network. Make sure that a semicolon is inserted at the end of each line, because otherwise `dhcpcd` is not started.

The sample file can be divided into three sections. The first one defines how many seconds an IP address is leased to a requesting client by default (`default-lease-`

time) before it should apply for renewal. This section also includes a statement of the maximum period for which a machine may keep an IP address assigned by the DHCP server without applying for renewal (`max-lease-time`).

In the second part, some basic network parameters are defined on a global level:

- The line `option domain-name` defines the default domain of your network.
- With the entry `option domain-name-servers`, specify up to three values for the DNS servers used to resolve IP addresses into hostnames and vice versa. Ideally, configure a name server on your machine or somewhere else in your network before setting up DHCP. That name server should also define a hostname for each dynamic address and vice versa. To learn how to configure your own name server, read *Chapitre 15, The Domain Name System* (p. 283).
- The line `option broadcast-address` defines the broadcast address the requesting client should use.
- With `option routers`, set where the server should send data packets that cannot be delivered to a host on the local network (according to the source and target host address and the subnet mask provided). In most cases, especially in smaller networks, this router is identical to the Internet gateway.
- With `option subnet-mask`, specify the netmask assigned to clients.

The last section of the file defines a network, including a subnet mask. To finish, specify the address range that the DHCP daemon should use to assign IP addresses to interested clients. In Exemple 16.1, « The Configuration File `/etc/dhcpd.conf` » (p. 312), clients may be given any address between `192.168.2.10` and `192.168.2.20` as well as `192.168.2.100` and `192.168.2.200`.

After editing these few lines, you should be able to activate the DHCP daemon with the command `rcdhcpd start`. It will be ready for use immediately. Use the command `rcdhcpd check-syntax` to perform a brief syntax check. If you encounter any unexpected problems with your configuration (the server aborts with an error or does not return `done` on start), you should be able to find out what has gone wrong by looking for information either in the main system log `/var/log/messages` or on console 10 (Ctrl + Alt + F10).

On a default openSUSE system, the DHCP daemon is started in a chroot environment for security reasons. The configuration files must be copied to the chroot environment

so the daemon can find them. Normally, there is no need to worry about this because the command `redhcpd start` automatically copies the files.

16.3.1 Clients with Fixed IP Addresses

DHCP can also be used to assign a predefined, static address to a specific client. Addresses assigned explicitly always take priority over dynamic addresses from the pool. A static address never expires in the way a dynamic address would, for example, if there were not enough addresses available and the server needed to redistribute them among clients.

To identify a client configured with a static address, `dhcpd` uses the hardware address (which is a globally unique, fixed numerical code consisting of six octet pairs) for the identification of all network devices (for example, `00:30:6E:08:EC:80`). If the respective lines, like the ones in Exemple 16.2, « Additions to the Configuration File » (p. 314), are added to the configuration file of Exemple 16.1, « The Configuration File `/etc/dhcpd.conf` » (p. 312), the DHCP daemon always assigns the same set of data to the corresponding client.

Exemple 16.2 *Additions to the Configuration File*

```
host jupiter {  
    hardware ethernet 00:30:6E:08:EC:80;  
    fixed-address 192.168.2.100;  
}
```

The name of the respective client (host *hostname*, here `jupiter`) is entered in the first line and the MAC address in the second line. On Linux hosts, find the MAC address with the command `ip link show` followed by the network device (for example, `eth0`). The output should contain something like

```
link/ether 00:30:6E:08:EC:80
```

In the preceding example, a client with a network card having the MAC address `00:30:6E:08:EC:80` is assigned the IP address `192.168.2.100` and the host-name `jupiter` automatically. The type of hardware to enter is `ethernet` in nearly all cases, although `token-ring`, which is often found on IBM systems, is also supported.

16.3.2 The openSUSE Version

To improve security, the openSUSE version of the ISC's DHCP server comes with the non-root/chroot patch by Ari Edelkind applied. This enables `dhcpd` to run with

the user ID `nobody` and run in a chroot environment (`/var/lib/dhcp`). To make this possible, the configuration file `dhcpd.conf` must be located in `/var/lib/dhcp/etc`. The init script automatically copies the file to this directory when starting.

Control the server's behavior regarding this feature by means of entries in the file `/etc/sysconfig/dhcpd`. To run `dhcpd` without the chroot environment, set the variable `DHCPD_RUN_CHROOTED` in `/etc/sysconfig/dhcpd` to « no ».

To enable `dhcpd` to resolve hostnames even from within the chroot environment, some other configuration files must be copied as well:

- `/etc/localtime`
- `/etc/host.conf`
- `/etc/hosts`
- `/etc/resolv.conf`

These files are copied to `/var/lib/dhcp/etc/` when starting the init script. Take these copies into account for any changes that they require if they are dynamically modified by scripts like `/etc/ppp/ip-up`. However, there should be no need to worry about this if the configuration file only specifies IP addresses (instead of hostnames).

If your configuration includes additional files that should be copied into the chroot environment, set these under the variable `DHCPD_CONF_INCLUDE_FILES` in the file `/etc/sysconfig/dhcpd`. To ensure that the DHCP logging facility keeps working even after a restart of the `syslog-ng` daemon, there is an additional entry `SYSLOGD_ADDITIONAL_SOCKET_DHCP` in the file `/etc/sysconfig/syslog`.

16.4 For More Information

More information about DHCP is available at the Web site of the *Internet Systems Consortium* (<http://www.isc.org/products/DHCP/>). Information is also available in the `dhcpd`, `dhcpd.conf`, `dhcpd.leases`, and `dhcp-options` man pages.

Time Synchronization with NTP

The NTP (network time protocol) mechanism is a protocol for synchronizing the system time over the network. First, a machine can obtain the time from a server that is a reliable time source. Second, a machine can itself act as a time source for other computers in the network. The goal is twofold—maintaining the absolute time and synchronizing the system time of all machines within a network.

Maintaining an exact system time is important in many situations. The built-in hardware (BIOS) clock does often not meet the requirements of applications such as databases or clusters. Manual correction of the system time would lead to severe problems because, for example, a backward leap can cause malfunction of critical applications. Within a network, it is usually necessary to synchronize the system time of all machines, but manual time adjustment is a bad approach. NTP provides a mechanism to solve these problems. The NTP service continuously adjusts the system time with the help of reliable time servers in the network. It further enables the management of local reference clocks, such as radio-controlled clocks.

17.1 Configuring an NTP Client with YaST

The NTP daemon (`ntpd`) coming with the `ntp` package is preset to use the local computer clock as a time reference. Using the (BIOS) clock, however, only serves as a fallback for cases where no time source of better precision is available. YaST facilitates the configuration of an NTP client.

17.1.1 Basic Configuration

The YaST NTP client configuration (*Network Services > NTP Configuration*) consists of tabs. Set the start mode of `ntpd` and the server to query on the *General Settings* tab.

Figure 17.1 *Advanced NTP Configuration: General Settings*

The screenshot shows the 'Advanced NTP Configuration' dialog box with the 'General Settings' tab selected. The title bar reads 'Advanced NTP Configuration' with a subtitle 'Select whether to start the NTP daemon now and on every system boot. [more](#)'. The dialog has two tabs: 'General Settings' and 'Security Settings'. Under 'Start NTP Daemon', there are three radio buttons: 'Only Manually', 'Synchronize without Daemon', and 'Now and On Boot' (which is selected). Below this is a 'Runtime Configuration Policy' section with a dropdown menu set to 'Auto' and a text field. The 'Interval of the Synchronization in Minutes' is set to 30. A table lists synchronization sources: 'Undisciplined Local Clock (LOCAL)' and 'Server 1.opensuse.pool.ntp.org' (which is highlighted). At the bottom are buttons for 'Add', 'Edit', 'Delete', and 'Display Log...'. The footer contains 'Help', 'Cancel', and 'OK' buttons.

Synchronization Type	Address
Undisciplined Local Clock (LOCAL)	
Server	1.opensuse.pool.ntp.org

Only Manually

Select *Only Manually*, if you want to configure everything on your own.

Synchronize without Daemon

On laptops and other machines that suspend automatically, select *Synchronize without Daemon*. Using this mode, YaST does not start `ntpd` on the local machine for synchronizing. Instead YaST creates a crontab entry (`/etc/cron.d/novell.ntp-synchronize`) that checks the time with the time server as specified in the *Interval of the Synchronization in Minutes* field. For more information about cron, see Section 11.1.2, « The cron Package » (p. 192).

Now and On Boot

Select *Now and On Boot* to start `ntpd` automatically when the system is booted. Either of `0.opensuse.pool.ntp.org`, `1.opensuse.pool.ntp.org`, `2.opensuse.pool.ntp.org`, or `3.opensuse.pool.ntp.org` is pre-selected.

17.1.2 Changing Basic Configuration

The servers and other time sources for the client to query are listed in the lower part of the *General Settings* tab. Modify this list as needed with *Add*, *Edit*, and *Delete*.

Display Log provides the possibility to view the log files of your client.

Click *Add* to add a new source of time information. In the following dialog, select the type of source with which the time synchronization should be made. The following options are available:

Figure 17.2 *YaST: NTP Server*



Server

In the pull-down *Select* list (see Figure 17.2, « YaST: NTP Server » (p. 319), determine whether to set up time synchronization using a time server from your

local network (*Local NTP Server*) or an Internet-based time server that takes care of your time zone (*Public NTP Server*). For a local time server, click *Lookup* to start an SLP query for available time servers in your network. Select the most suitable time server from the list of search results and exit the dialog with *OK*. For a public time server, select your country (time zone) and a suitable server from the list under *Public NTP Server* then exit the dialog with *OK*. In the main dialog, test the availability of the selected server with *Test*. *Options* allows you to specify additional options for `ntpd`.

Using *Access Control Options*, you can restrict the actions that the remote computer can perform with the daemon running on your computer. This field is enabled only after checking *Restrict NTP Service to Configured Servers Only* on the *Security Settings* tab (see Figure 17.3, « Advanced NTP Configuration: Security Settings » (p. 321)). The options correspond to the `restrict` clauses in `/etc/ntp.conf`. For example, `nomodify notrap noquery` disallows the server to modify NTP settings of your computer and to use the trap facility (a remote event logging feature) of your NTP daemon. Using these restrictions is recommended for servers out of your control (for example, on the Internet).

Refer to `/usr/share/doc/packages/ntp-doc` (part of the `ntp-doc` package) for detailed information.

Peer

A peer is a machine to which a symmetric relationship is established: it acts both as a time server and as a client. To use a peer in the same network instead of a server, enter the address of the system. The rest of the dialog is identical to the *Server* dialog.

Radio Clock

To use a radio clock in your system for the time synchronization, enter the clock type, unit number, device name, and other options in this dialog. Click *Driver Calibration* to fine-tune the driver. Detailed information about the operation of a local radio clock is available in `/usr/share/doc/packages/ntp-doc/refclock.html`.

Outgoing Broadcast

Time information and queries can also be transmitted by broadcast in the network. In this dialog, enter the address to which such broadcasts should be sent. Do not activate broadcasting unless you have a reliable time source like a radio controlled clock.

Incoming Broadcast

If you want your client to receive its information via broadcast, enter the address from which the respective packets should be accepted in this fields.

Figure 17.3 *Advanced NTP Configuration: Security Settings*



In the *Security Settings* tab (see Figure 17.3, « Advanced NTP Configuration: Security Settings » (p. 321)), determine whether `ntpd` should be started in a chroot jail. By default, *Run NTP Daemon in Chroot Jail* is activated. This increases the security in the event of an attack over `ntpd`, as it prevents the attacker from compromising the entire system.

Restrict NTP Service to Configured Servers Only increases the security of your system by disallowing remote computers to view and modify NTP settings of your computer and to use the trap facility for remote event logging. Once enabled, these restrictions apply to all remote computers, unless you override the access control options for individual computers in the list of time sources in the *General Settings* tab. For all other remote computers, only querying for local time is allowed.

Enable *Open Port in Firewall* if `SuSEfirewall2` is active (which it is by default). If you leave the port closed, it is not possible to establish a connection to the time server.

17.2 Manually Configuring ntp in the Network

The easiest way to use a time server in the network is to set server parameters. For example, if a time server called `ntp.example.com` is reachable from the network, add its name to the file `/etc/ntp.conf` by adding the following line:

```
server ntp.example.com
```

To add more time servers, insert additional lines with the keyword `server`. After initializing `ntpd` with the command `rcntp start`, it takes about one hour until the time is stabilized and the drift file for correcting the local computer clock is created. With the drift file, the systematic error of the hardware clock can be computed as soon as the computer is powered on. The correction is used immediately, resulting in a higher stability of the system time.

There are two possible ways to use the NTP mechanism as a client: First, the client can query the time from a known server in regular intervals. With many clients, this approach can cause a high load on the server. Second, the client can wait for NTP broadcasts sent out by broadcast time servers in the network. This approach has the disadvantage that the quality of the server is unknown and a server sending out wrong information can cause severe problems.

If the time is obtained via broadcast, you do not need the server name. In this case, enter the line `broadcastclient` in the configuration file `/etc/ntp.conf`. To use one or more known time servers exclusively, enter their names in the line starting with `servers`.

17.3 Dynamic Time Synchronization at Runtime

If the system boots without network connection, `ntpd` starts up, but it cannot resolve DNS names of the time servers set in the configuration file. This can happen if you use Network Manager with an encrypted WLAN.

If you want `ntpd` to resolve DNS names at runtime, you must set the `dynamic` option. Then, when the network is established some time after booting, `ntpd` looks up the names again and can reach the time servers to get the time.

Manually edit `/etc/ntp.conf` and add `dynamic` to one or more server entries:

```
server ntp.example.com dynamic
```

Or use YaST and proceed as follows:

- 1 In YaST click *Network Services > NTP Configuration*.
- 2 Select the server you want to configure. Then click *Edit*.
- 3 Activate the *Options* field and add `dynamic`. Separate it with a space, if there are already other options entered.
- 4 Click *Ok* to close the edit dialog. Repeat the previous step to change all servers as wanted.
- 5 Finally click *Ok* to save the settings.

17.4 Setting Up a Local Reference Clock

The software package `ntp` contains drivers for connecting local reference clocks. A list of supported clocks is available in the `ntp-doc` package in the file `/usr/share/doc/packages/ntp-doc/refclock.html`. Every driver is associated with a number. In `ntp`, the actual configuration takes place by means of pseudo IP addresses. The clocks are entered in the file `/etc/ntp.conf` as though they existed in the network. For this purpose, they are assigned special IP addresses in the form `127.127.t.u`. Here, `t` stands for the type of the clock and determines which driver is used and `u` for the unit, which determines the interface used.

Normally, the individual drivers have special parameters that describe configuration details. The file `/usr/share/doc/packages/ntp-doc/drivers/driverNN.html` (where `NN` is the number of the driver) provides information about the particular type of clock. For example, the « type 8 » clock (radio clock over serial interface) requires an additional mode that specifies the clock more precisely. The Conrad DCF77 receiver module, for example, has mode 5. To use this clock as a preferred reference, specify the keyword `prefer`. The complete `server` line for a Conrad DCF77 receiver module would be:

```
server 127.127.8.0 mode 5 prefer
```

Other clocks follow the same pattern. Following the installation of the `ntp-doc` package, the documentation for `ntp` is available in the directory `/usr/share/doc/packages/ntp-doc`. The file `/usr/share/doc/packages/ntp-doc/refclock.html` provides links to the driver pages describing the driver parameters.

Sharing File Systems with NFS

Distributing and sharing file systems over a network is a common task in corporate environments. The well-proven network file system (*NFS*) works together with *NIS*, the yellow pages protocol. For a more secure protocol that works together with *LDAP* and may also use Kerberos, check *NFSv4*.

NFS with NIS makes a network transparent to the user. With NFS, it is possible to distribute arbitrary file systems over the network. With an appropriate setup, users always find themselves in the same environment regardless of the terminal they currently use.

IMPORTANT : Need for DNS

In principle, all exports can be made using IP addresses only. To avoid time-outs, you need a working DNS system. DNS is necessary at least for logging purposes, because the mountd daemon does reverse lookups.

18.1 Terminology

The following are terms used in the YaST module.

Exports

A directory *exported* by an NFS server, which clients can integrate it into their system.

NFS Client

The NFS client is a system that uses NFS services from an NFS server over the Network File System protocol. The TCP/IP protocol is already integrated into the Linux kernel; there is no need to install any additional software.

NFS Server

The NFS server provides NFS services to clients. A running server depends on the following daemons: `nfsd` (worker), `idmapd` (user and group name mappings to IDs and vice versa), `statd` (file locking), and `mountd` (mount requests).

18.2 Installing NFS Server

The NFS server software is not part of the default installation. If you configure an NFS Server as described in Section 18.3, « Configuring NFS Server » (p. 326) you will automatically be prompted to install the needed packages. Alternatively, install the package `nfs-kernel-server` with YaST or zypper.

Like NIS, NFS is a client/server system. However, a machine can be both—it can supply file systems over the network (export) and mount file systems from other hosts (import).

18.3 Configuring NFS Server

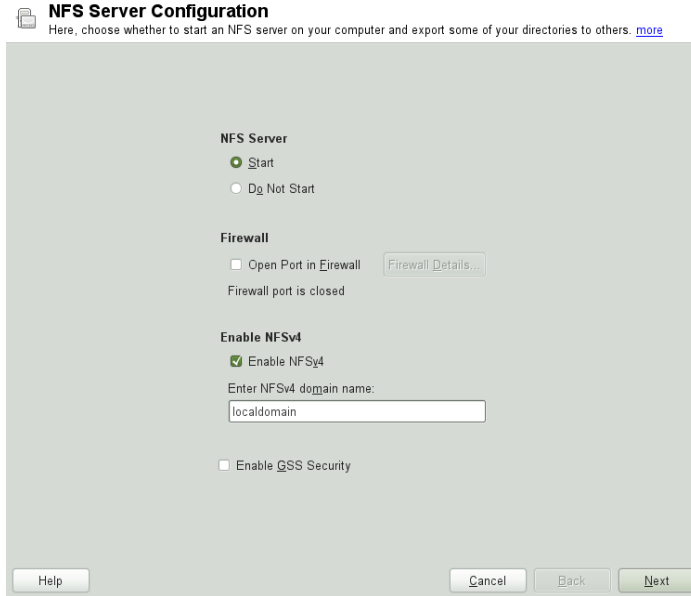
Configuring an NFS server can be done either through YaST or manually. For authentication, NFS can also be combined with Kerberos.

18.3.1 Exporting File Systems with YaST

With YaST, turn a host in your network into an NFS server—a server that exports directories and files to all hosts granted access to it. The server can also provide applications to all members of a group without installing the applications locally on each and every host. To set up such a server, proceed as follows:

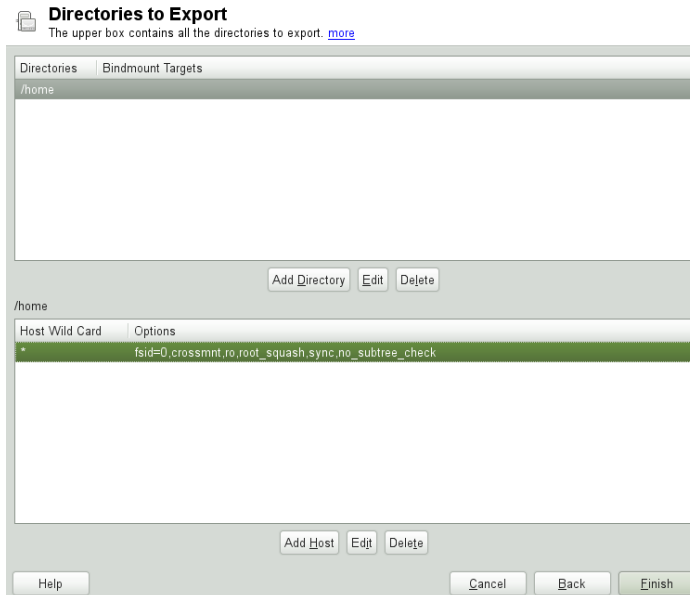
- 1 Start YaST and select *Network Services > NFS Server*; see Figure 18.1, « NFS Server Configuration Tool » (p. 327).

Figure 18.1 *NFS Server Configuration Tool*



- 2 Activate the *Start* radio button and enter the *NFSv4 Domain Name*.
- 3 Click *Enable GSS Security* if you need secure access to the server. A prerequisite for this is to have Kerberos installed on your domain and to have both the server and the clients kerberized. Click *Next*.
- 4 Enter the directories to export in the upper text field. Below, enter the hosts that should have access to them. This dialog is shown in Figure 18.2, « Configuring an NFS Server with YaST » (p. 328).

Figure 18.2 *Configuring an NFS Server with YaST*



The figure shows the scenario where NFSv4 is enabled in the previous dialog. Bindmount Targets is shown in the right pane. For more details, click *Help*. In the lower half of the dialog, there are four options that can be set for each host: single host, netgroups, wildcards, and IP networks. For a more thorough explanation of these options, refer to the `exports` man page.

- 5 Click *Finish* to complete the configuration.

IMPORTANT : Automatic Firewall Configuration

If a firewall is active on your system (SuSEfirewall2), YaST adapts its configuration for the NFS server by enabling the `nfs` service when *Open Ports in Firewall* is selected.

18.3.1.1 Exporting for NFSv4 Clients

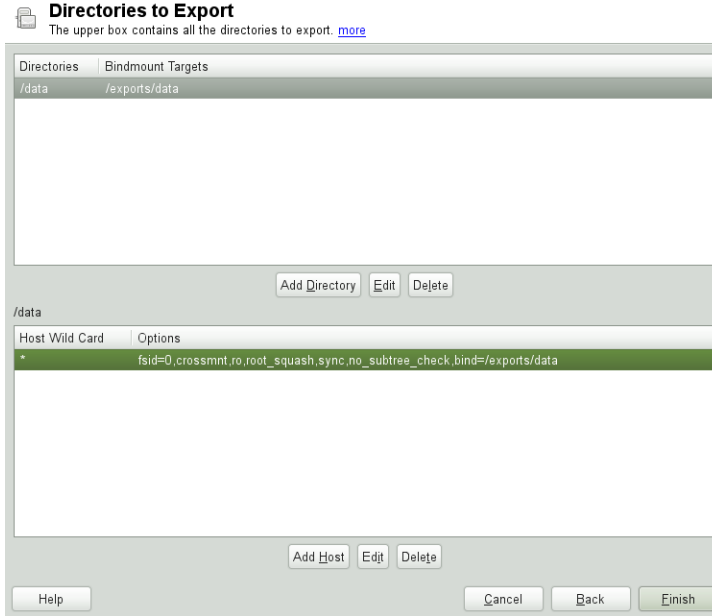
Activate *Enable NFSv4* to support NFSv4 clients. Clients with NFSv3 can still access the server's exported directories if they are exported appropriately. This is explained in detail in Section 18.3.1.3, « Coexisting v3 and v4 Exports » (p. 332).

After activating NFSv4, enter an appropriate domain name. Make sure the name is the same as the one in the `/etc/idmapd.conf` file of any NFSv4 client that accesses this particular server. This parameter is for the `idmapd` service that is required for NFSv4 support (on both server and client). Leave it as `localdomain` (the default) if you do not have special requirements. For more information, see the links in Section 18.5, « For More Information » (p. 338).

Click *Next*. The dialog that follows has two sections. The upper half consists of two columns named *Directories* and *Bind Mount Targets*. *Directories* is a directly editable column that lists the directories to export.

For a fixed set of clients, there are two types of directories that can be exported—directories that act as pseudo root file systems and those that are bound to some subdirectory of the pseudo file system. This pseudo file system acts as a base point under which all file systems exported for the same client set take their place. For a client or set of clients, only one directory on the server can be configured as pseudo root for export. For this client, export multiple directories by binding them to some existing subdirectory in the pseudo root.

Figure 18.3 *Exporting Directories with NFSv4*



In the lower half of the dialog, enter the client (wild card) and export options for a particular directory. After adding a directory in the upper half, another dialog for entering the client information and options pops up automatically. After that, to add a new client or a set of clients, click *Add Host*.

In the small dialog that opens, enter the host wild card. There are four possible types of host wild cards that can be set for each host: a single host (name or IP address), netgroups, wild cards (such as `*` indicating all machines can access the server), and IP networks. Then, in *Options*, include `fsid=0` in the comma-separated list of options to configure the directory as pseudo root. If this directory needs to be bound to another directory under an already configured pseudo root, make sure that a target bind path is given in the option list with `bind=/target/path`.

For example, suppose that the directory `/exports` is chosen as the pseudo root directory for all the clients that can access the server. Then add this in the upper half and make sure that the options entered for this directory include `fsid=0`. If there is another directory, `/data`, that also needs to be NFSv4 exported, add this directory to the upper half. While entering options for this, make sure that `bind=/exports/data` is in the list and that `/exports/data` is an already existing subdirectory of

/exports. Any change in the option `bind=/target/path`, whether addition, deletion, or change in value, is reflected in *Bindmount Targets*. This column is not a directly editable column, but instead summarizes directories and their nature. After all information is provided, click *Finish* to complete the configuration. The service will become available immediately.

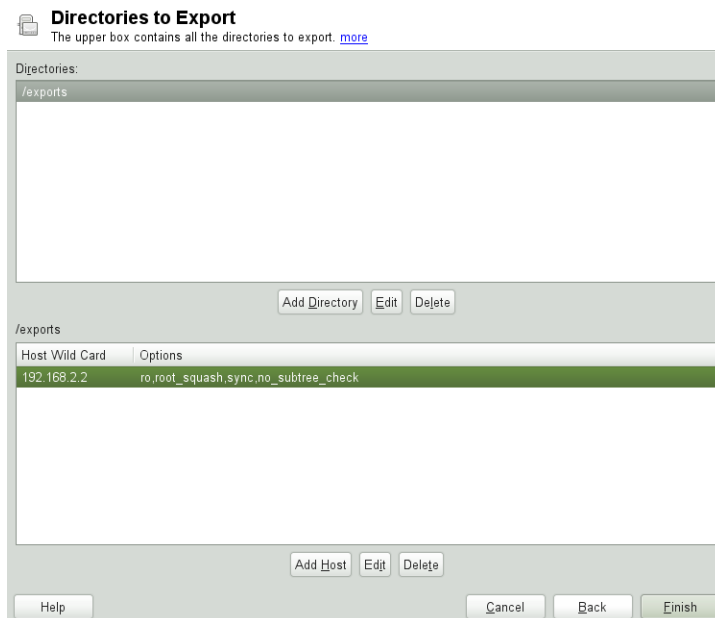
18.3.1.2 NFSv3 and NFSv2 Exports

Make sure that *Enable NFSv4* is not checked in the initial dialog before clicking *Next*.

The next dialog has two parts. In the upper text field, enter the directories to export. Below, enter the hosts that should have access to them. There are four types of host wild cards that can be set for each host: a single host (name or IP address), netgroups, wild cards (such as `*` indicating all machines can access the server), and IP networks.

This dialog is shown in Figure 18.4, « Exporting Directories with NFSv2 and v3 » (p. 331). Find a more thorough explanation of these options in `man exports`. Click *Finish* to complete the configuration.

Figure 18.4 *Exporting Directories with NFSv2 and v3*



18.3.1.3 Coexisting v3 and v4 Exports

NFSv3 and NFSv4 exports can coexist on a server. After enabling the support for NFSv4 in the initial configuration dialog, those exports for which `fsid=0` and `bind=/target/path` are not included in the option list are considered v3 exports. Consider the example in Figure 18.2, « Configuring an NFS Server with YaST » (p. 328). If you add another directory, such as `/data2`, using *Add Directory* then in the corresponding options list do not mention either `fsid=0` or `bind=/target/path`, this export acts as a v3 export.

IMPORTANT

Automatic Firewall Configuration

If `SUSEfirewall2` is active on your system, YaST adapts its configuration for the NFS server by enabling the `nfs` service when *Open Ports in Firewall* is selected.

18.3.2 Exporting File Systems Manually

The configuration files for the NFS export service are `/etc/exports` and `/etc/sysconfig/nfs`. In addition to these files, `/etc/idmapd.conf` is needed for the NFSv4 server configuration. To start or restart the services, run the command `rcnfsserver restart`. This also starts the `rpc.idmapd` if NFSv4 is configured in `/etc/sysconfig/nfs`. The NFS server depends on a running RPC portmapper. Therefore, also start or restart the portmapper service with `rcrpcbind restart`.

18.3.2.1 Exporting File Systems with NFSv4

NFSv4 is the latest version of NFS protocol available on openSUSE. Configuring directories for export with NFSv4 differs slightly from previous NFS versions.

`/etc/exports`

The `/etc/exports` file contains a list of entries. Each entry indicates a directory that is shared and how it is shared. A typical entry in `/etc/exports` consists of:

```
/shared/directory    host(option_list)
```

For example:

```
/export 192.168.1.2(rw,fsid=0,sync,crossmnt)
/export/data 192.168.1.2(rw,bind=/data,sync)
```

Here the IP address `192.168.1.2` is used to identify the allowed client. You can also use the name of the host, a wild card indicating a set of hosts (`*.abc.com`, `*`, etc.), or netgroups (`@my-hosts`).

The directory which specifies `fsid=0` is special. It is the root of the filesystem that is exported, sometimes referred to as the pseudo root filesystem. This directory must also have the `crossmnt` for correct operation with NFSv4. All other directories exported via NFSv4 must be mounted below this point. If you want to export a directory that is not within the exported root, it needs to be bound into the exported tree. This can be done using the `bind=` syntax.

In the example above, `/data` is not within `/export`, so we export `/export/data`, and specify that the `/data` directory should be bound to that name. The directory `/export/data` must exist and should normally be empty.

When clients mount from this server, they just mount `servername:/` rather than `servername:/export`. It is not necessary to mount `servername:/data`, because it will automatically appear beneath wherever `servername:/` was mounted.

/etc/sysconfig/nfs

The `/etc/sysconfig/nfs` file contains a few parameters that determine NFSv4 server daemon behavior. It is important to set the parameter `NFS4_SUPPORT` to `yes`. `NFS4_SUPPORT` determines whether the NFS server supports NFSv4 exports and clients.

/etc/idmapd.conf

Every user on a Linux machine has a name and ID. `idmapd` does the name-to-ID mapping for NFSv4 requests to the server and replies to the client. It must be running on both server and client for NFSv4, because NFSv4 uses only names for its communication.

Make sure that there is a uniform way in which usernames and IDs (uid) are assigned to users across machines that might probably be sharing file systems using NFS. This

can be achieved by using NIS, LDAP, or any uniform domain authentication mechanism in your domain.

The parameter `Domain` must be set the same for both, client and server in the `/etc/idmapd.conf` file. If you are not sure, leave the domain as `localdomain` in the server and client files. A sample configuration file looks like the following:

```
[General]

Verbosity = 0
Pipefs-Directory = /var/lib/nfs/rpc_pipefs
Domain = localdomain

[Mapping]

Nobody-User = nobody
Nobody-Group = nobody
```

For further reference, read the man page of `idmapd` and `idmapd.conf`; `man idmapd`, `man idmapd.conf`.

Starting and Stopping Services

After changing `/etc/exports` or `/etc/sysconfig/nfs`, start or restart the NFS server service with `rcnfsserver restart`. After changing `/etc/idmapd.conf`, reload the configuration file with the command `killall -HUP rpc.idmapd`.

If the NFS service needs to start at boot time, run the command `chkconfig nfsserver on`.

18.3.2.2 Exporting File Systems with NFSv2 and NFSv3

This section is specific to NFSv3 and NFSv2 exports. Refer to Section 18.3.1.1, « Exporting for NFSv4 Clients » (p. 329) for exporting with NFSv4.

Exporting file systems with NFS involves two configuration files: `/etc/exports` and `/etc/sysconfig/nfs`. A typical `/etc/exports` file entry is in the format:

```
/shared/directory    host(list_of_options)
```

For example:


```
/export 192.168.1.2(rw, sync)
```

Here, the directory `/export` is shared with the host `192.168.1.2` with the option `list rw, sync`. This IP address can be replaced with a client name or set of clients using a wild card (such as `*.abc.com`) or even `netgroups`.

For a detailed explanation of all options and their meaning, refer to the man page of `exports` (`man exports`).

After changing `/etc/exports` or `/etc/sysconfig/nfs`, start or restart the NFS server using the command `rcnfsserver restart`.

18.3.3 NFS with Kerberos

To use Kerberos authentication for NFS, GSS security must be enabled. Select *Enable GSS Security* in the initial YaST NFS Server dialog. You must have a working Kerberos server to use this feature. YaST does not set up the server but just uses the provided functionality. If you want to use Kerberos authentication in addition to the YaST configuration, complete at least the following steps before running the NFS configuration:

- 1 Make sure that both the server and the client are in the same Kerberos domain. They must access the same KDC (Key Distribution Center) server and share their `krb5.keytab` file (the default location on any machine is `/etc/krb5.keytab`). For more information about Kerberos, see Chapitre 6, *Network Authentication with Kerberos* (↑Security Guide).
- 2 Start the `gssd` service on the client with `rcgssd start`.
- 3 Start the `svcgssd` service on the server with `rcsvcgssd start`.

For more information about configuring kerberized NFS, refer to the links in Section 18.5, « For More Information » (p. 338).

18.4 Configuring Clients

To configure your host as an NFS client, you do not need to install additional software. All needed packages are installed by default.

18.4.1 Importing File Systems with YaST

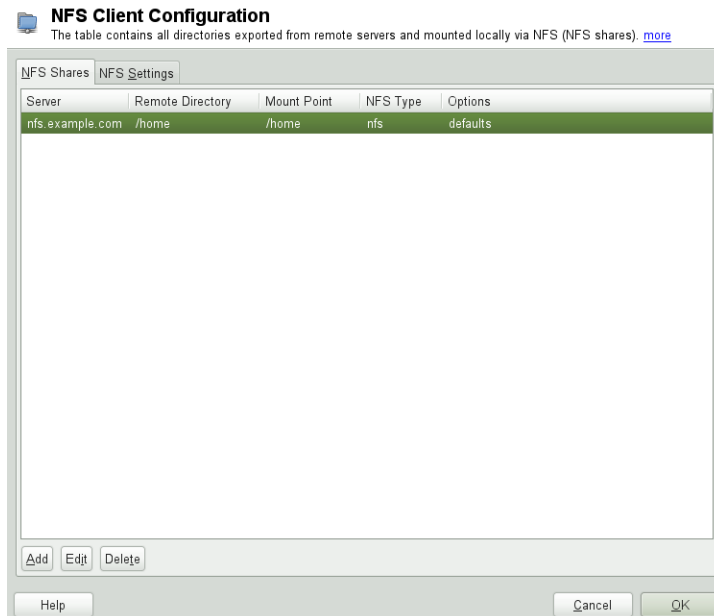
Authorized users can mount NFS directories from an NFS server into the local file tree using the YaST NFS client module. Click on *Add* and enter the hostname of the NFS server, the directory to import, and the mount point at which to mount this directory locally. The changes will take effect after clicking *Finish* in the first dialog.

In the *NFS Settings* tab, enable *Open Port in Firewall* to allow access to the service from remote computers. The firewall status is displayed next to the check box. When using NFSv4, make sure that the checkbox *Enable NFSv4* is selected and that the *NFSv4 Domain Name* contains the same value as used by the NFSv4 server. The default domain is `localdomain`.

Click *OK* to save your changes. See Figure 18.5, « NFS Client Configuration with YaST » (p. 336).

The configuration is written to `/etc/fstab` and the specified file systems are mounted. When you start the YaST configuration client at a later time, it also reads the existing configuration from this file.

Figure 18.5 *NFS Client Configuration with YaST*



18.4.2 Importing File Systems Manually

The prerequisite for importing file systems manually from an NFS server is a running RPC port mapper. Start it by entering `rpcbind start` as root. Then remote file systems can be mounted in the file system like local partitions using `mount`:

```
mount host:remote-path local-path
```

To import user directories from the `nfs.example.com` machine, for example, use:

```
mount nfs.example.com:/home /home
```

18.4.2.1 Using the Automount Service

The `autofs` daemon can be used to mount remote file systems automatically. Add the following entry in the your `/etc/auto.master` file:

```
/nfsmounts /etc/auto.nfs
```

Now the `/nfsmounts` directory acts as the root for all the NFS mounts on the client if the `auto.nfs` file is filled appropriately. The name `auto.nfs` is chosen for the sake of convenience—you can choose any name. In `auto.nfs` add entries for all the NFS mounts as follows:

```
localdata -fstype=nfs server1:/data
nfs4mount -fstype=nfs4 server2:/
```

Activate the settings with `rcautofs start` as root. In this example, `/nfsmounts/localdata`, the `/data` directory of `server1`, is mounted with NFS and `/nfsmounts/nfs4mount` from `server2` is mounted with NFSv4.

If the `/etc/auto.master` file is edited while the service `autofs` is running, the automounter must be restarted for the changes to take effect with `rcautofs restart`.

18.4.2.2 Manually Editing `/etc/fstab`

A typical NFSv3 mount entry in `/etc/fstab` looks like this:

```
nfs.example.com:/data /local/path nfs rw,noauto 0 0
```

NFSv4 mounts may also be added to the `/etc/fstab` file. For these mounts, use `nfs4` instead of `nfs` in the third column and make sure that the remote file system is

given as `/` after the `nfs.example.com:` in the first column. A sample line for an NFSv4 mount in `/etc/fstab` looks like this:

```
nfs.example.com:/ /local/pathv4 nfs4 rw,noauto 0 0
```

The `noauto` option prevents the file system from being mounted automatically at start up. If you want to mount the respective file system manually, it is possible to shorten the mount command specifying the mount point only:

```
mount /local/path
```

Note, that if you do not enter the `noauto` option, the initialization scripts of the system will handle the mount of those file systems at start up.

18.5 For More Information

In addition to the man pages of `exports`, `nfs`, and `mount`, information about configuring an NFS server and client is available in `/usr/share/doc/packages/nfsidmap/README`. For further documentation online refer to the following Web sites:

- Find the detailed technical documentation online at SourceForge [<http://nfs.sourceforge.net/>].
- For instructions for setting up kerberized NFS, refer to NFS Version 4 Open Source Reference Implementation [<http://www.citi.umich.edu/projects/nfsv4/linux/krb5-setup.html>].
- If you have questions on NFSv4, refer to the Linux NFSv4 FAQ [<http://www.citi.umich.edu/projects/nfsv4/linux/faq/>].

Samba

Using Samba, a Unix machine can be configured as a file and print server for Mac OS X, Windows, and OS/2 machines. Samba has developed into a fully-fledged and rather complex product. Configure Samba with YaST, SWAT (a Web interface), or by editing the configuration file manually.

19.1 Terminology

The following are some terms used in Samba documentation and in the YaST module.

SMB protocol

Samba uses the SMB (server message block) protocol that is based on the NetBIOS services. Microsoft released the protocol so other software manufacturers could establish connections to a Microsoft domain network. With Samba, the SMB protocol works on top of the TCP/IP protocol, so the TCP/IP protocol must be installed on all clients.

CIFS protocol

CIFS (common Internet file system) protocol is another protocol supported by Samba. CIFS defines a standard remote file system access protocol for use over the network, enabling groups of users to work together and share documents across the network.

NetBIOS

NetBIOS is a software interface (API) designed for communication between machines providing a name service. It enables machines connected to the network

to reserve names for themselves. After reservation, these machines can be addressed by name. There is no central process that checks names. Any machine on the network can reserve as many names as it wants as long as the names are not already in use. The NetBIOS interface can be implemented for different network architectures. An implementation that works relatively closely with network hardware is called NetBEUI, but this is often referred to as NetBIOS. Network protocols implemented with NetBIOS are IPX from Novell (NetBIOS via TCP/IP) and TCP/IP.

The NetBIOS names sent via TCP/IP have nothing in common with the names used in `/etc/hosts` or those defined by DNS. NetBIOS uses its own, completely independent naming convention. However, it is recommended to use names that correspond to DNS hostnames to make administration easier or use DNS natively. This is the default used by Samba.

Samba server

Samba server provides SMB/CIFS services and NetBIOS over IP naming services to clients. For Linux, there are three daemons for Samba server: `smbd` for SMB/CIFS services, `nmbd` for naming services, and `winbind` for authentication.

Samba client

The Samba client is a system that uses Samba services from a Samba server over the SMB protocol. All common operating systems, such as Mac OS X, Windows, and OS/2, support the SMB protocol. The TCP/IP protocol must be installed on all computers. Samba provides a client for the different UNIX flavors. For Linux, there is a kernel module for SMB that allows the integration of SMB resources on the Linux system level. You do not need to run any daemon for the Samba client.

Shares

SMB servers provide resources to the clients by means of shares. Shares are printers and directories with their subdirectories on the server. It is exported by means of a name and can be accessed by its name. The share name can be set to any name—it does not have to be the name of the export directory. A printer is also assigned a name. Clients can access the printer by its name.

DC

A domain controller (DC) is a server that handles accounts in domain. For data replication, additional domain controllers are available in one domain.

19.2 Installing a Samba Server

To install a Samba server, start YaST and select *Software > Software Management*. Choose *Filter > Patterns* and select *File Server*. Confirm the installation of the required packages to finish the installation process.

19.3 Starting and Stopping Samba

You can start or stop the Samba server automatically (during boot) or manually. Starting and stopping policy is a part of the YaST Samba server configuration described in Section 19.4.1, « Configuring a Samba Server with YaST » (p. 341).

To stop or start running Samba services with YaST, use *System > System Services (Runlevel)* and check winbind, smb, and nmb. From a command line, stop services required for Samba with `rcsmb stop && rcnmb stop` and start them with `rcnmb start && rcsmb start`; rcsmb cares about winbind if needed.

19.4 Configuring a Samba Server

A Samba server in openSUSE® can be configured in two different ways: with YaST or manually. Manual configuration offers a higher level of detail, but lacks the convenience of the YaST GUI.

19.4.1 Configuring a Samba Server with YaST

To configure a Samba server, start YaST and select *Network Services > Samba Server*.

19.4.1.1 Initial Samba Configuration

When starting the module for the first time, the *Samba Installation* dialog starts, prompting you to make just a few basic decisions concerning administration of the server. At the end of the configuration it prompts for the Samba administrator password (*Samba Root Password*). For later starts, the *Samba Configuration* dialog appears.

The *Samba Installation* dialog consists of two steps and optional detailed settings:

Workgroup or Domain Name

Select an existing name from *Workgroup or Domain Name* or enter a new one and click *Next*.

Samba Server Type

In the next step, specify whether your server should act as a primary domain controller (PDC), backup domain controller (BDC), or not to act as a domain controller at all. Continue with *Next*.

If you do not want to proceed with a detailed server configuration, confirm with *OK*. Then in the final pop-up box, set the *Samba root Password*.

You can change all settings later in the *Samba Configuration* dialog with the *Start-Up*, *Shares*, *Identity*, *Trusted Domains*, and *LDAP Settings* tabs.

19.4.1.2 Advanced Samba Configuration

During the first start of the Samba server module the *Samba Configuration* dialog appears directly after the two initial steps described in Section 19.4.1.1, « Initial Samba Configuration » (p. 341). Use it to adjust your Samba server configuration.

After editing your configuration, click *OK* to save your settings.

Starting the Server

In the *Start Up* tab, configure the start of the Samba server. To start the service every time your system boots, select *During Boot*. To activate manual start, choose *Manually*. More information about starting a Samba server is provided in Section 19.3, « Starting and Stopping Samba » (p. 341).

In this tab, you can also open ports in your firewall. To do so, select *Open Port in Firewall*. If you have multiple network interfaces, select the network interface for Samba services by clicking *Firewall Details*, selecting the interfaces, and clicking *OK*.

Shares

In the *Shares* tab, determine the Samba shares to activate. There are some predefined shares, like homes and printers. Use *Toggle Status* to switch between *Active* and *Inactive*. Click *Add* to add new shares and *Delete* to delete the selected share.

Allow Users to Share Their Directories enables members of the group in *Permitted Group* to share directories they own with other users. For example, `users` for a local scope or `DOMAIN\Users` for a domain scope. The user also must make sure that the file system permissions allow access. With *Maximum Number of Shares*, limit the total amount of shares that may be created. To permit access to user shares without authentication, enable *Allow Guest Access*.

Identity

In the *Identity* tab, you can determine the domain with which the host is associated (*Base Settings*) and whether to use an alternative hostname in the network (*NetBIOS Hostname*). It is also possible to use Microsoft Windows Internet Name Service (WINS) for name resolution. In this case, activate *Use WINS for Hostname Resolution* and decide whether to *Retrieve WINS server via DHCP*. To set expert global settings or set a user authentication source, click *Advanced Settings*.

Trusted Domains

To enable users from other domains to access your domain, make the appropriate settings in the *Trusted Domains* tab. To add a new domain, click *Add*. To remove the selected domain, click *Delete*.

LDAP Settings

In the tab *LDAP Settings*, you can determine the LDAP server to use for authentication. To test the connection to your LDAP server, click *Test Connection*. To set expert LDAP settings or use default values, click *Advanced Settings*.

For more information about LDAP configuration, see Chapitre 4, *LDAP—A Directory Service* (↑Security Guide).

19.4.2 Web Administration with SWAT

An alternative tool for Samba server administration is SWAT (Samba Web Administration Tool). It provides a simple Web interface with which to configure the Samba server. To use SWAT, open <http://localhost:901> in a Web browser and log in as user `root`. If you do not have a special Samba root account, use the system `root` account.

REMARQUE : Activating SWAT

After Samba server installation, SWAT is not activated. To activate it, open *Network Services > Network Services (xinetd)* in YaST, enable the network services configuration, select *swat* from the table, and click *Toggle Status (On or Off)*.

19.4.3 Configuring the Server Manually

If you intend to use Samba as a server, install `samba`. The main configuration file for Samba is `/etc/samba/smb.conf`. This file can be divided into two logical parts. The `[global]` section contains the central and global settings. The `[share]` sections contain the individual file and printer shares. By means of this approach, details regarding the shares can be set differently or globally in the `[global]` section, which enhances the structural transparency of the configuration file.

19.4.3.1 The global Section

The following parameters of the `[global]` section need some adjustment to match the requirements of your network setup so other machines can access your Samba server via SMB in a Windows environment.

```
workgroup = TUX-NET
```

This line assigns the Samba server to a workgroup. Replace `TUX-NET` with an appropriate workgroup of your networking environment. Your Samba server appears under its DNS name unless this name has been assigned to some other machine in the network. If the DNS name is not available, set the server name using `netbiosname=MYNAME`. For more details about this parameter, see the `smb.conf` man page.

```
os level = 20
```

This parameter triggers whether your Samba server tries to become LMB (local master browser) for its workgroup. With the Samba 3 release series, it is seldom necessary to override the default setting (20). Choose a very low value such as 2 to spare the existing Windows network from any disturbances caused by a misconfigured Samba server. More information about this important topic can be found in the Network Browsing chapter of the Samba 3 Howto; for more information on the Samba 3 Howto, see Section 19.7, « For More Information » (p. 349).

If no other SMB server is present in your network (such as a Windows 2000 server) and you want the Samba server to keep a list of all systems present in the local environment, set the `os_level` to a higher value (for example, 65). Your Samba server is then chosen as LMB for your local network.

When changing this setting, consider carefully how this could affect an existing Windows network environment. First test the changes in an isolated network or at a noncritical time of day.

`wins support` and `wins server`

To integrate your Samba server into an existing Windows network with an active WINS server, enable the `wins server` option and set its value to the IP address of that WINS server.

If your Windows machines are connected to separate subnets and need to still be aware of each other, you need to set up a WINS server. To turn a Samba server into such a WINS server, set the option `wins support = Yes`. Make sure that only one Samba server of the network has this setting enabled. The options `wins server` and `wins support` must never be enabled at the same time in your `smb.conf` file.

19.4.3.2 Shares

The following examples illustrate how a CD-ROM drive and the user directories (homes) are made available to the SMB clients.

[cdrom]

To avoid having the CD-ROM drive accidentally made available, these lines are deactivated with comment marks (semicolons in this case). Remove the semicolons in the first column to share the CD-ROM drive with Samba.

Example 19.1 *A CD-ROM Share (deactivated)*

```
[cdrom]
;      comment = Linux CD-ROM
;      path = /media/cdrom
;      locking = No
```

[cdrom] and comment

The `[cdrom]` section entry is the name of the share that can be seen by all SMB clients on the network. An additional `comment` can be added to further describe the share.

```
path = /media/cdrom
path exports the directory /media/cdrom.
```

By means of a very restrictive default configuration, this kind of share is only made available to the users present on this system. If this share should be made available to everybody, add a line `guest ok = yes` to the configuration. This setting gives read permissions to anyone on the network. It is recommended to handle this parameter with great care. This applies even more to the use of this parameter in the `[global]` section.

[homes]

The `[homes]` share is of special importance here. If the user has a valid account and password for the Linux file server and his own home directory, he can be connected to it.

Example 19.2 *[homes] Share*

```
[homes]
comment = Home Directories
valid users = %S
browseable = No
read only = No
create mask = 0640
directory mask = 0750
```

[homes]

As long as there is no other share using the share name of the user connecting to the SMB server, a share is dynamically generated using the `[homes]` share directives. The resulting name of the share is the username.

```
valid users = %S
```

`%S` is replaced with the concrete name of the share as soon as a connection has been successfully established. For a `[homes]` share, this is always the username. As a consequence, access rights to a user's share are restricted exclusively to that user.

```
browseable = No
```

This setting makes the share invisible in the network environment.

```
read only = No
```

By default, Samba prohibits write access to any exported share by means of the `read only = Yes` parameter. To make a share writable, set the value `read only = No`, which is synonymous with `writable = Yes`.

```
create mask = 0640
```

Systems that are not based on MS Windows NT do not understand the concept of UNIX permissions, so they cannot assign permissions when creating a file. The parameter `create mask` defines the access permissions assigned to newly created files. This only applies to writable shares. In effect, this setting means the owner has read and write permissions and the members of the owner's primary group have read permissions. `valid users = %S` prevents read access even if the group has read permissions. For the group to have read or write access, deactivate the line `valid users = %S`.

19.4.3.3 Security Levels

To improve security, each share access can be protected with a password. SMB offers the following ways of checking permissions:

Share Level Security (`security = share`)

A password is firmly assigned to a share. Everyone who knows this password has access to that share.

User Level Security (`security = user`)

This variant introduces the concept of the user to SMB. Each user must register with the server with his or her own password. After registration, the server can grant access to individual exported shares dependent on usernames.

Server Level Security (`security = server`)

To its clients, Samba pretends to be working in user level mode. However, it passes all password queries to another user level mode server, which takes care of authentication. This setting requires the additional `password server` parameter.

ADS Level Security (`security = ADS`)

In this mode, Samba will act as a domain member in an Active Directory environment. To operate in this mode, the machine running Samba needs Kerberos installed and configured. You must join the machine using Samba to the ADS realm. This can be done using the YaST *Windows Domain Membership* module.

Domain Level Security (`security = domain`)

This mode will only work correctly if the machine has been joined into a Windows NT Domain. Samba will try to validate username and password by passing it to a Windows NT Primary or Backup Domain Controller. The same

way as a Windows NT Server would do. It expects the encrypted passwords parameter to be set to `yes`.

The selection of share, user, server, or domain level security applies to the entire server. It is not possible to offer individual shares of a server configuration with share level security and others with user level security. However, you can run a separate Samba server for each configured IP address on a system.

More information about this subject can be found in the Samba 3 HOWTO. For multiple servers on one system, pay attention to the options `interfaces` and `bind interfaces only`.

19.5 Configuring Clients

Clients can only access the Samba server via TCP/IP. NetBEUI and NetBIOS via IPX cannot be used with Samba.

19.5.1 Configuring a Samba Client with YaST

Configure a Samba client to access resources (files or printers) on the Samba or Windows server. Enter the NT or Active Directory domain or workgroup in the dialog *Network Services > Windows Domain Membership*. If you activate *Also Use SMB Information for Linux Authentication*, the user authentication runs over the Samba, NT or Kerberos server.

Click *Expert Settings* for advanced configuration options. For example, use the *Mount Server Directories* table to enable mounting server home directory automatically with authentication. This way users will be able to access their home directories when hosted on CIFS. For details, see the `pam_mount` man page.

After completing all settings, confirm the dialog to finish the configuration.

19.6 Samba as Login Server

In networks where predominantly Windows clients are found, it is often preferable that users may only register with a valid account and password. In a Windows-based

network, this task is handled by a primary domain controller (PDC). You can use a Windows NT server configured as PDC, but this task can also be done with a Samba server. The entries that must be made in the `[global]` section of `smb.conf` are shown in Example 19.3, « Global Section in `smb.conf` » (p. 349).

Example 19.3 *Global Section in `smb.conf`*

```
[global]
    workgroup = TUX-NET
    domain logons = Yes
    domain master = Yes
```

If encrypted passwords are used for verification purposes the Samba server must be able to handle these. The entry `encrypt passwords = yes` in the `[global]` section enables this (with Samba version 3, this is now the default). In addition, it is necessary to prepare user accounts and passwords in an encryption format that conforms with Windows. Do this with the command `smbpasswd -a name`. Create the domain account for the computers, required by the Windows domain concept, with the following commands:

```
useradd hostname\$$
smbpasswd -a -m hostname
```

With the `useradd` command, a dollar sign is added. The command `smbpasswd` inserts this automatically when the parameter `-m` is used. The commented configuration example (`/usr/share/doc/packages/samba/examples/smb.conf.SUSE`) contains settings that automate this task.

```
add machine script = /usr/sbin/useradd -g nogroup -c "NT Machine Account" \
-s /bin/false %m\$$
```

To make sure that Samba can execute this script correctly, choose a Samba user with the required administrator permissions and add it to the `ntadmin` group. Then all users belonging to this Linux group can be assigned `Domain Admin` status with the command:

```
net groupmap add ntgroup="Domain Admins" unixgroup=ntadmin
```

For more information about this topic, see Chapter 12 of the Samba 3 HOWTO, found in `/usr/share/doc/packages/samba/Samba3-HOWTO.pdf`.

19.7 For More Information

Detailed Samba information is available in the digital documentation. Enter `apropos samba` at the command line to display some manual pages or just browse

the `/usr/share/doc/packages/samba` directory if Samba documentation is installed for more online documentation and examples. Find a commented example configuration (`smb.conf.SUSE`) in the `examples` subdirectory.

The Samba 3 HOWTO provided by the Samba team includes a section about troubleshooting. In addition to that, Part V of the document provides a step-by-step guide to checking your configuration. You can find Samba 3 HOWTO in `/usr/share/doc/packages/samba/Samba3-HOWTO.pdf` after installing the package `samba-doc`.

Also read the Samba page in the openSUSE wiki at <http://en.opensuse.org/Samba>.

Le serveur HTTP Apache

Avec une part de marché de plus de 50%, le serveur HTTP Apache est le serveur web le plus utilisé dans le monde selon l'enquête de <http://www.netcraft.com/>. Apache, développé par l'Apache Software Foundation (<http://www.apache.org/>), est disponible pour la plupart des systèmes d'exploitation. openSUSE® inclut la version 2.2 d'Apache. Dans ce chapitre, apprenez comment installer, configurer et mettre en place un serveur web ; comment utiliser SSL, CGI, et les modules additionnels ; et comment régler les problèmes d'Apache.

20.1 Démarrage rapide

Grâce à cette section, mettez en place et démarrez rapidement Apache. Vous devez être `root` pour installer et configurer Apache.

20.1.1 Requirements

Assurez-vous que les conditions suivantes sont remplies avec d'essayer de mettre en place le serveur web Apache :

1. Le réseau de la machine est correctement configuré. Pour plus d'informations à ce sujet, référez-vous à Chapitre 13, *Basic Networking* (p. 219).
2. Le système de temps exact de la machine est maintenu par synchronisation avec un serveur de temps. Cela est nécessaire car des parties du protocole HTTP dépendent

de l'heure exacte. Voir Chapitre 17, *Time Synchronization with NTP* (p. 317) pour en apprendre davantage à ce sujet.

3. Les dernières mises à jour de sécurité sont installées. En cas de doute, lancez YaST Online Update.
4. The default Web server port (80) is opened in the firewall. For this, configure the SuSEFirewall2 to allow the service *HTTP Server* in the external zone. This can be done using YaST. See section « Configuring the Firewall with YaST » (Chapitre 13, *Masquerading and Firewalls*, ↑Security Guide) for details.

20.1.2 Installation

Apache n'est pas installé par défaut sur openSUSE. Pour l'installer avec une configuration standard prédéfinie qui fonctionne « out of the box », procédez comme ci-dessous :

Procédure 20.1 *Installation d'Apache avec la configuration par défaut*

- 1 Démarrez YaST et sélectionnez *Logiciel > Installer et supprimer des logiciels*.
- 2 Choisissez *Vue > Schémas* et sélectionnez *Serveur Web et LAMP* dans *Fonctions de serveur*.
- 3 Confirmez l'installation des paquets nécessaires pour terminer le processus d'installation.

L'installation comprend le module de multitraitement `apache2-prefork` ainsi que le module PHP5. Reportez-vous à Section 20.4, « Installation, activation et configuration de modules » (p. 371) pour plus d'informations sur ces modules.

20.1.3 Démarrer

Vous pouvez lancer Apache automatiquement au démarrage ou le lancer manuellement.

Procédure 20.2 *Démarrage automatique d'Apache*

- 1 Pour vous assurer qu'Apache est automatiquement lancé au démarrage en niveaux d'exécution 3 et 5, exécutez la commande suivante :

```
chkconfig -a apache2
```

- 2 Vous pouvez également démarrer YaST et sélectionner *Système > Services système (niveaux d'exécution)*.

- 3 Search for *apache2* and *Enable* the service.

Le serveur web démarre immédiatement.

- 4 Enregistrez vos modifications avec *Terminer*.

Le système est configuré pour lancer Apache automatiquement en niveaux d'exécution 3 et 5 au démarrage.

Pour démarrer manuellement Apache avec le terminal, tapez `rcapache2 start`.

Procédure 20.3 *Vérifier si Apache est en cours d'exécution*

Si vous ne recevez pas de message d'erreur en lançant Apache, cela signifie généralement que le serveur web fonctionne. Pour tester cela :

- 1 Démarrez un navigateur et ouvrez <http://localhost/>.

Si Apache est installé et fonctionnel, vous obtenez une page de test indiquant « It works! ».

- 2 Si vous ne voyez pas cette page, référez-vous à Section 20.8, « Troubleshooting » (p. 390).

Maintenant que le serveur web fonctionne, vous pouvez ajouter vos propres documents, ajuster la configuration selon vos besoins, ou ajouter des fonctionnalités en installant des modules.

20.2 Configuration d'Apache

openSUSE offre deux options de configuration :

- Configurer Apache manuellement (p. 357)
- Configuring Apache with YaST (p. 362)

La configuration manuelle offre un haut niveau de détails, mais il lui manque la commodité de l'interface graphique YaST.

IMPORTANT : Rechargez ou relancez Apache après les modifications de configuration

La plupart des changements de configuration nécessitent le rechargement d'Apache (parfois également un redémarrage) pour prendre effet. Rechargez manuellement Apache avec `rcapache2 reload` ou utilisez l'une des options de redémarrage décrite dans Section 20.3, « Démarrage et arrêt d'Apache » (p. 368).

Si vous configurez Apache avec YaST, cela peut être pris en charge automatiquement si vous passez *Service HTTP* à *Activé* comme décrit dans Section 20.2.3.2, « HTTP Server Configuration » (p. 366).

20.2.1 Fichiers de configuration d'Apache

Cette section donne un aperçu des fichiers de configuration d'Apache. Si vous utilisez YaST pour la configuration, vous n'avez pas besoin de toucher à ces fichiers—cependant, ces informations peuvent vous être utiles si vous souhaitez passer à une configuration manuelle plus tard.

Les fichiers de configuration d'Apache peuvent être trouvés dans deux endroits différents :

- `/etc/sysconfig/apache2` (p. 354)
- `/etc/apache2/` (p. 355)

20.2.1.1 `/etc/sysconfig/apache2`

`/etc/sysconfig/apache2` contrôle certains paramètres généraux d'Apache, comme les modules à charger, les fichiers de configuration additionnels à inclure, les options avec lesquelles le serveur doit être démarré, et les options qui doivent être ajoutées à la ligne de commande. Chaque option de configuration de ce fichier est abondamment documentée et n'est donc pas mentionnée ici. Pour un serveur web classique, les paramètres de `/etc/sysconfig/apache2` devraient être suffisants pour n'importe quel besoin de configuration.

20.2.1.2 /etc/apache2/

/etc/apache2/ accueille tous les fichiers de configuration pour Apache. Dans ce qui suit, l'utilité de chaque fichier est expliquée. Chaque fichier comprend plusieurs options de configuration (également appelées *directives*). Chaque option de configuration de ces fichiers est largement documentée et n'est donc pas mentionnée ici.

Les fichiers de configuration d'Apache sont organisés comme ceci :

```
/etc/apache2/
|
|- charset.conv
|- conf.d/
|   |
|   |- *.conf
|
|- default-server.conf
|- errors.conf
|- httpd.conf
|- listen.conf
|- magic
|- mime.types
|- mod_*.conf
|- server-tuning.conf
|- ssl.*
|- ssl-global.conf
|- sysconfig.d
|   |
|   |- global.conf
|   |- include.conf
|   |- loadmodule.conf . .
|
|- uid.conf
|- vhosts.d
|   |- *.conf
```

Fichiers de configuration d'Apache dans /etc/apache2/

`charset.conv`

Spécifie le jeu de caractères à utiliser pour différentes langues. Ne modifiez pas ce fichier.

`conf.d/*.conf`

Fichiers de configuration ajoutés par d'autres modules. Ces fichiers de configuration peuvent être inclus dans votre configuration de virtualhost si nécessaire. Voir `vhosts.d/vhost.template` pour des exemples. En procédant ainsi, vous pouvez fournir des ensembles de modules différents pour différents virtualhosts.

`default-server.conf`

Configuration globale pour tous les virtualhosts avec des valeurs par défaut raisonnables. Au lieu de changer les valeurs, écrasez-les avec une configuration de virtualhost.

`errors.conf`

Définit comment Apache répond aux erreurs. Pour personnaliser ces messages pour tous les virtualhosts, éditez ce fichier. Sinon, écrasez ces directives dans vos configurations de virtualhost.

`httpd.conf`

Le principal fichier de configuration d'Apache. Évitez de modifier ce fichier. Il contient principalement les instructions `include` et les paramètres globaux. Écrasez les paramètres globaux dans les fichiers de configuration correspondants listés ici. Changez les paramètres spécifiques à l'hôte (comme la racine des documents) dans la configuration de votre virtualhost.

`listen.conf`

Lie Apache à des adresses IP et des ports spécifiques. L'hébergement virtuel par nom est également configuré ici. Pour plus de détails, consultez la section intitulée « Virtualhosts basés sur le nom » (p. 359).

`magic`

Données pour le module `mime_magic` qui permet à Apache de déterminer automatiquement le type MIME d'un fichier inconnu. Ne modifiez pas ce fichier.

`mime.types`

Types MIME reconnus par le système (il s'agit en fait d'un lien vers `/etc/mime.types`). Ne modifiez pas ce fichier. Si vous avez besoin d'ajouter des types MIME qui ne figurent pas ici, ajoutez-les dans `mod_mime-defaults.conf`.

`mod_*.conf`

Fichiers de configuration pour les modules qui sont installés par défaut. Reportez-vous à Section 20.4, « Installation, activation et configuration de modules » (p. 371) pour plus de détails. Notez que les fichiers de configuration pour les modules optionnels se situent dans le répertoire `conf.d`.

`server-tuning.conf`

Contient les directives de configuration pour les différents MPMs (voir Section 20.4.4, « Modules multi-processus » (p. 376)) ainsi que les options de

configuration générale qui contrôlent les performances d'Apache. Testez correctement votre serveur web lorsque vous faites des changements ici.

`ssl-global.conf` et `ssl.*`

Configuration SSL globale et données de certificat SSL. Consultez Section 20.6, « Setting Up a Secure Web Server with SSL » (p. 381) pour plus de détails.

`sysconfig.d/*.conf`

Fichiers de configuration générés automatiquement à partir de `/etc/sysconfig/apache2`. Ne modifiez aucun de ces fichiers—éditez `/etc/sysconfig/apache2` à la place. Ne mettez pas d'autres fichiers de configuration dans ce répertoire.

`uid.conf`

Spécifie sous quel utilisateur et quel ID de groupe Apache s'exécute. Ne modifiez pas ce fichier.

`vhosts.d/*.conf`

La configuration de votre virtualhost doit se trouver ici. Le répertoire contient des modèles de fichier pour les virtualhosts avec et sans SSL. Chaque fichier de ce répertoire se terminant par `.conf` est automatiquement inclus dans la configuration d'Apache. Reportez-vous à Section 20.2.2.1, « Configuration de virtualhost » (p. 357) pour plus de détails.

20.2.2 Configurer Apache manuellement

La configuration manuelle d'Apache nécessite la modification des fichiers textes de configuration en tant qu'utilisateur `root`.

20.2.2.1 Configuration de virtualhost

Le terme *virtualhost* fait référence à la capacité d'Apache de servir plusieurs URIs (Universal Resource Identifiers) depuis la même machine physique. Cela signifie que plusieurs domaines, comme `www.example.com` et `www.example.net`, sont gérés par un seul serveur web sur une seule machine physique.

Il est courant d'utiliser des virtualhosts pour réduire les tâches d'administration (un seul serveur web doit être maintenu) et les frais de matériel (chaque domaine ne nécessite pas un serveur dédié). Les virtualhosts peuvent être basés sur le nom, l'IP, ou le port.

Pour lister tous les virtualhosts existants, utilisez la commande `httpd2 -S`. Cela affiche une liste indiquant le serveur par défaut et tous les virtualhosts ainsi que leurs adresses IP et ports d'écoute. En outre, la liste contient aussi une entrée pour chaque virtualhost indiquant son emplacement dans les fichiers de configuration.

Les virtualhosts peuvent être configurés avec YaST comme décrit dans la section intitulée « Virtual Hosts » (p. 365) ou en éditant manuellement un fichier de configuration. Par défaut, Apache est préparé dans openSUSE pour utiliser un fichier de configuration par virtualhost dans `/etc/apache2/vhosts.d/`. Tous les fichiers de ce répertoire possédant l'extension `.conf` sont automatiquement inclus dans la configuration. Un modèle de virtualhost basique est fourni dans ce répertoire (`vhost.template` ou `vhost-ssl.template` pour un virtualhost avec le support de SSL).

ASTUCE : Toujours créer une configuration de virtualhost

Il est recommandé de toujours créer un fichier de configuration de virtualhost, même si votre serveur web n'héberge qu'un seul domaine. En faisant cela, vous avez non seulement la configuration spécifique au domaine dans un fichier, mais vous pouvez également toujours revenir à une configuration fonctionnelle de base en déplaçant, supprimant ou renommant simplement le fichier de configuration du virtualhost. Pour la même raison, vous devez également créer des fichiers de configuration séparés pour chaque virtualhost.

Lors de l'utilisation de virtualhosts basés sur le nom, il est recommandé de mettre en place une configuration par défaut qui sera utilisée quand un nom de domaine ne correspond pas à une configuration de virtualhost. Le virtualhost par défaut est celui dont la configuration est chargée en premier. Étant donné que l'ordre des fichiers de configuration est déterminé par le nom de fichier, débutez le nom du fichier de configuration du virtualhost par défaut avec un caractère underscore (`_`) pour vous assurer qu'il soit chargé en premier (par exemple : `_default_vhost.conf`).

Le bloc `<VirtualHost></VirtualHost>` comprend les informations qui s'appliquent à un domaine particulier. Quand Apache reçoit la demande d'un client pour un virtualhost défini, il utilise les directives incluses dans cette section. Presque toutes les directives peuvent être utilisées dans le contexte d'un virtualhost. Pour plus d'informations sur les directives de configuration d'Apache, consultez <http://httpd.apache.org/docs/2.2/mod/quickreference.html>.

Virtualhosts basés sur le nom

Avec des virtualhosts basés sur le nom, plusieurs sites web peuvent être desservis par adresse IP. Apache utilise le champ d'hôte dans l'en-tête HTTP qui est envoyée par le client pour connecter la requête à une entrée `ServerName` correspondante dans l'une des déclarations de `virtualhost`. Si aucun `ServerName` correspondant n'est trouvé, le premier virtualhost spécifié est utilisé comme valeur par défaut.

La directive `NameVirtualHost` indique à Apache sur quelle adresse IP, et éventuellement, quel port il doit écouter pour les requêtes de clients contenant le nom de domaine dans l'en-tête HTTP. Cette option est configurée dans le fichier de configuration `/etc/apache2/listen.conf`.

Le premier argument peut être un nom de domaine pleinement qualifié (FQDN, Fully Qualified Domain Name), mais il est recommandé d'utiliser l'adresse IP. Le second argument est le port, mais il est optionnel. Par défaut, le port 80 est utilisé et configuré via la directive `Listen`.

Le joker `*` peut être utilisé aussi bien pour l'adresse IP que pour le numéro de port afin de recevoir les requêtes sur toutes les interfaces. Les adresses IPv6 doivent être définies entre crochets.

Exemple 20.1 *Variations of Name-Based VirtualHost Entries*

```
# NameVirtualHost Adresse IP[:Port]
NameVirtualHost 192.168.3.100:80
NameVirtualHost 192.168.3.100
NameVirtualHost *:80
NameVirtualHost *
NameVirtualHost [2002:c0a8:364::]:80
```

The opening `VirtualHost` tag takes the IP address (or fully qualified domain name) previously declared with the `NameVirtualHost` as an argument in a name-based virtual host configuration. A port number previously declared with the `NameVirtualHost` directive is optional.

Le joker `*` est aussi autorisé comme substitut pour l'adresse IP. Cette syntaxe n'est valable qu'en association avec l'utilisation du joker dans `NameVirtualHost *`. Lorsque vous utilisez des adresses IPv6, l'adresse doit figurer entre crochets.

Exemple 20.2 *Directives VirtualHost basées sur le nom*

```
<VirtualHost 192.168.3.100:80>
...
</VirtualHost>
```

```

<VirtualHost 192.168.3.100>
    ...
</VirtualHost>

<VirtualHost *:80>
    ...
</VirtualHost>

<VirtualHost *>
    ...
</VirtualHost>

<VirtualHost [2002:c0a8:364::]>
    ...
</VirtualHost>

```

Virtualhosts basés sur l'IP

Cette configuration de virtualhost alternative nécessite la configuration de plusieurs adresses IP pour une machine. Une seule instance d'Apache héberge plusieurs domaines, dont chacun se voit attribuer une IP différente.

Le serveur physique doit avoir une adresse IP pour chaque virtualhost basé sur l'IP. Si la machine ne dispose pas de plusieurs cartes réseau, des interfaces réseau virtuelles (IP aliasing) peuvent également être utilisées.

L'exemple suivant montre Apache fonctionnant sur une machine avec l'IP 192.168.3.100, hébergeant deux domaines sur les IP additionnelles 192.168.3.101 et 192.168.3.102. Un bloc `VirtualHost` séparé est nécessaire pour chaque serveur virtuel.

Exemple 20.3 *Directives VirtualHost basées sur l'IP*

```

<VirtualHost 192.168.3.101>
    ...
</VirtualHost>

<VirtualHost 192.168.3.102>
    ...
</VirtualHost>

```

Ici, les directives `VirtualHost` sont spécifiées uniquement pour les interfaces autres que 192.168.3.100. Quand une directive `Listen` est également configurée pour 192.168.3.100, un hôte virtuel distinct basé sur l'IP doit être créé pour répondre aux requêtes HTTP vers cette interface—dans le cas contraire, les directives trouvées dans la configuration par défaut du serveur (`/etc/apache2/default-server.conf`) sont appliquées.

Configuration de virtualhost basique

Au minimum, les directives suivantes doivent être présentes dans chaque configuration de virtualhost afin de mettre en place un virtualhost. Consultez `/etc/apache2/vhosts.d/vhost.template` pour plus d'informations.

`ServerName`

The fully qualified domain name under which the host should be addressed.

`DocumentRoot`

Path to the directory from which Apache should serve files for this host. For security reasons, access to the entire file system is forbidden by default, so you must explicitly unlock this directory within a `Directory` container.

`ServerAdmin`

Adresse e-mail de l'administrateur du serveur. Cette adresse apparaît, par exemple, sur les pages d'erreur créées par Apache.

`ErrorLog`

The error log file for this virtual host. Although it is not necessary to create separate error log files for each virtual host, it is common practice to do so, because it makes the debugging of errors much easier. `/var/log/apache2/` is the default directory for Apache's log files.

`CustomLog`

The access log file for this virtual host. Although it is not necessary to create separate access log files for each virtual host, it is common practice to do so, because it allows the separate analysis of access statistics for each host. `/var/log/apache2/` is the default directory for Apache's log files.

Comme mentionné ci-dessus, l'accès à l'intégralité du système de fichiers est interdit par défaut pour des raisons de sécurité. Par conséquent, déverrouillez explicitement les répertoires dans lesquels vous avez placé les fichiers qu'Apache doit servir—par exemple le `DocumentRoot` :

```
<Directory "/srv/www/www.example.com/docs">
    Order allow,deny
    Allow from all
</Directory>
```

Le fichier de configuration complet ressemble à ceci :

Exemple 20.4 *Configuration VirtualHost basique*

```
<VirtualHost 192.168.3.100>
```

```

ServerName www.example.com
DocumentRoot /srv/www/www.example.com/htdocs
ServerAdmin webmaster@example.com
ErrorLog /var/log/apache2/www.example.com_log
CustomLog /var/log/apache2/www.example.com-access_log common
<Directory "/srv/www/www.example.com/htdocs">
    Order allow,deny
    Allow from all
</Directory>
</VirtualHost>

```

20.2.3 Configuring Apache with YaST

To configure your Web server with YaST, start YaST and select *Network Services > HTTP Server*. When starting the module for the first time, the *HTTP Server Wizard* starts, prompting you to make a few basic decisions concerning administration of the server. After having finished the wizard, the *HTTP Server Configuration* dialog starts each time you call the *HTTP Server* module. For more information, see Section 20.2.3.2, « HTTP Server Configuration » (p. 366).

20.2.3.1 HTTP Server Wizard

The HTTP Server Wizard consists of five steps. In the last step of the dialog, you are given the opportunity to enter the expert configuration mode to make even more specific settings.

Network Device Selection

Here, specify the network interfaces and ports Apache uses to listen for incoming requests. You can select any combination of existing network interfaces and their respective IP addresses. Ports from all three ranges (well-known ports, registered ports, and dynamic or private ports) that are not reserved by other services can be used. The default setting is to listen on all network interfaces (IP addresses) on port 80.

Check *Open Port In Firewall* to open the ports in the firewall that the Web server listens on. This is necessary to make the Web server available on the network, which can be a LAN, WAN, or the public Internet. Keeping the port closed is only useful in test situations where no external access to the Web server is necessary. If you have multiple network interfaces, click *Firewall Details...* to specify on which interface(s) the port(s) should be opened.

Click *Next* to continue with the configuration.

Modules


The *Modules* configuration option allows for the activation or deactivation of the script languages that the Web server should support. For the activation or deactivation of other modules, refer to la section intitulée « Server Modules » (p. 367). Click *Next* to advance to the next dialog.

Default Host

This option pertains to the default Web server. As explained in Section 20.2.2.1, « Configuration de virtualhost » (p. 357), Apache can serve multiple virtual hosts from a single physical machine. The first declared virtual host in the configuration file is commonly referred to as the *default host*. Each virtual host inherits the default host's configuration.

To edit the host settings (also called *directives*), choose the appropriate entry in the table then click *Edit*. To add new directives, click *Add*. To delete a directive, select it and click *Delete*.

Figure 20.1 HTTP Server Wizard: Default Host

 **HTTP Server Wizard (3/5)--Default Host**
To edit the host settings, choose the appropriate entry of the table then click Edit. [more](#)

Option	Value
Document Root	/usr/www/htdocs
Directory	/usr/www/htdocs/...
Alias	/icons/ /usr/share/apache2/icons/
Directory	/usr/share/apache2/icons/...
ScriptAlias	/cgi-bin/ /usr/www/cgi-bin/
Directory	/usr/www/cgi-bin/...
mod_userdir.c	
Include	/etc/apache2/conf.d/* conf
Include	/etc/apache2/conf.d/apache2-manual?conf
Server Name	archimedes
Server Administrator E-Mail	root@archimedes

Here is list of the default settings of the server:

Document Root

Path to the directory from which Apache serves files for this host. `/srv/www/htdocs` is the default location.

Alias

With the help of `Alias` directives, URLs can be mapped to physical file system locations. This means that a certain path even outside the `Document Root` in the file system can be accessed via a URL aliasing that path.

The default openSUSE `Alias /icons` points to `/usr/share/apache2/icons` for the Apache icons displayed in the directory index view.

ScriptAlias

Similar to the `Alias` directive, the `ScriptAlias` directive maps a URL to a file system location. The difference is that `ScriptAlias` designates the target directory as a CGI location, meaning that CGI scripts should be executed in that location.

Directory

With `Directory` settings, you can enclose a group of configuration options that will only apply to the specified directory.

Access and display options for the directories `/srv/www/htdocs`, `/usr/share/apache2/icons` and `/srv/www/cgi-bin` are configured here. It should not be necessary to change the defaults.

Include

With `include`, additional configuration files can be specified. Two `Include` directives are already preconfigured: `/etc/apache2/conf.d/` is the directory containing the configuration files that come with external modules. With this directive, all files in this directory ending in `.conf` are included. With the second directive, `/etc/apache2/conf.d/apache2-manual.conf`, the `apache2-manual` configuration file is included.

Server Name

This specifies the default URL used by clients to contact the Web server. Use a fully qualified domain name (FQDN) to reach the Web server at `http://FQDN/` or its IP address. You cannot choose an arbitrary name here—the server must be « known » under this name.

Server Administrator E-Mail

E-mail address of the server administrator. This address is, for example, shown on error pages Apache creates.

After finishing with the *Default Host* step, click *Next* to continue with the configuration.

Virtual Hosts

In this step, the wizard displays a list of already configured virtual hosts (see Section 20.2.2.1, « Configuration de virtualhost » (p. 357)). If you have not made manual changes prior to starting the YaST HTTP wizard, no virtual host is present.

To add a host, click *Add* to open a dialog in which to enter basic information about the host, such as *Server Name*, *Server Contents Root* (`DocumentRoot`), and the *Administrator E-Mail*. *Server Resolution* is used to determine how a host is identified (name based or IP based). Specify the name or IP address with *Change Virtual Host ID*

Clicking *Next* advances to the second part of the virtual host configuration dialog.

In part two of the virtual host configuration you can specify whether or not to enable CGI scripts and which directory to use for these scripts. It is also possible to enable SSL. If you do so, you must specify the path to the certificate as well. See Section 20.6.2, « Configuring Apache with SSL » (p. 386) for details on SSL and certificates. With the *Directory Index* option, you can specify which file to display when the client requests a directory (by default, `index.html`). Add one or more filenames (space-separated) if you want to change this. With *Enable Public HTML*, the content of the users public directories (`~user/public_html/`) is made available on the server under `http://www.example.com/~user`.

IMPORTANT : Creating Virtual Hosts

It is not possible to add virtual hosts at will. If using name-based virtual hosts, each hostname must be resolved on the network. If using IP-based virtual hosts, you can assign only one host to each IP address available.

Summary

This is the final step of the wizard. Here, determine how and when the Apache server is started: when booting or manually. Also see a short summary of the configuration

made so far. If you are satisfied with your settings, click *Finish* to complete configuration. If you want to change something, click *Back* until you have reached the desired dialog. Clicking *HTTP Server Expert Configuration* opens the dialog described in Section 20.2.3.2, « HTTP Server Configuration » (p. 366).

Figure 20.2 *HTTP Server Wizard: Summary*



20.2.3.2 HTTP Server Configuration

The *HTTP Server Configuration* dialog also lets you make even more adjustments to the configuration than the wizard (which only runs if you configure your Web server for the first time). It consists of four tabs described in the following. No configuration option you change here is effective immediately—you always must confirm your changes with *Finish* to make them effective. Clicking *Abort* leaves the configuration module and discards your changes.

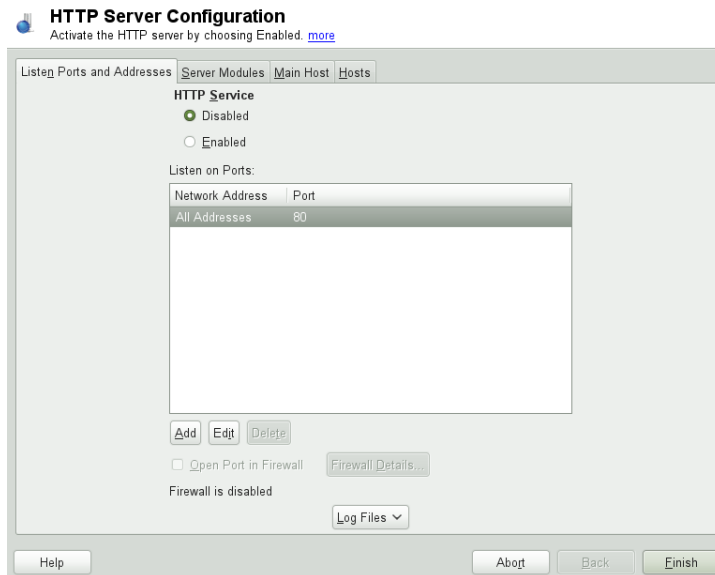
Listen Ports and Addresses

In *HTTP Service*, select whether Apache should be running (*Enabled*) or stopped (*Disabled*). In *Listen on Ports*, *Add*, *Edit*, or *Delete* addresses and ports on which the

server should be available. The default is to listen on all interfaces on port 80. You should always check *Open Port In Firewall*, because otherwise the Web server is not reachable from outside. Keeping the port closed is only useful in test situations where no external access to the Web server is necessary. If you have multiple network interfaces, click *Firewall Details...* to specify on which interface(s) the port(s) should be opened.

With *Log Files*, watch either the access log or the error log. This is useful if you want to test your configuration. The log file opens in a separate window from which you can also restart or reload the Web server. For details, see Section 20.3, « Démarrage et arrêt d'Apache » (p. 368). These commands are effective immediately and their log messages are also displayed immediately.

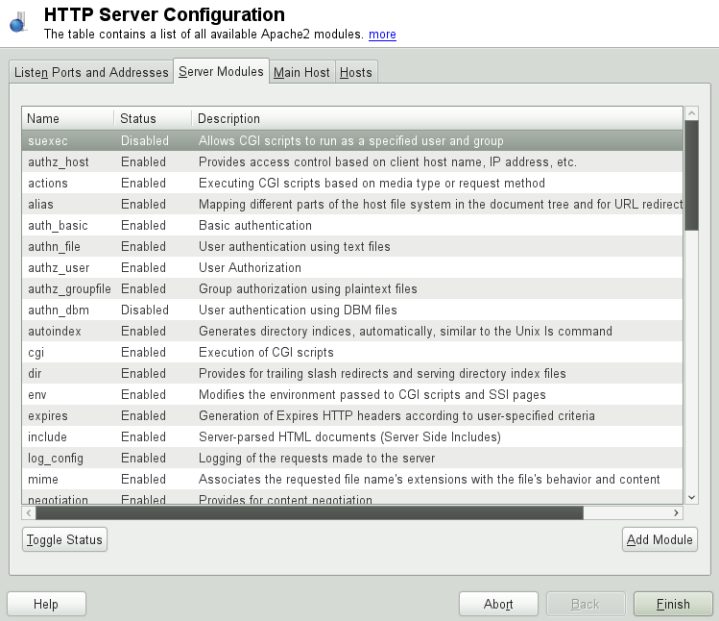
Figure 20.3 *HTTP Server Configuration: Listen Ports and Addresses*



Server Modules

You can change the status (enabled or disabled) of Apache2 modules by clicking *Toggle Status*. Click *Add Module* to add a new module that is already installed but not yet listed. Learn more about modules in Section 20.4, « Installation, activation et configuration de modules » (p. 371).

Figure 20.4 *HTTP Server Configuration: Server Modules*



Main Host or Hosts

These dialogs are identical to the ones already described. Refer to la section intitulée « Default Host » (p. 363) and la section intitulée « Virtual Hosts » (p. 365).

20.3 Démarrage et arrêt d'Apache

S'il est configuré avec YaST comme décrit dans Section 20.2.3, « Configuring Apache with YaST » (p. 362), Apache est lancé au démarrage aux niveaux d'exécution 3 et 5, et il est stoppé aux niveaux d'exécution 0, 1, 2 et 6. Vous pouvez modifier ce comportement en utilisant l'éditeur de niveaux d'exécution de YaST ou la ligne de commande `chkconfig`.

Pour démarrer, stopper, ou manipuler Apache sur un système en fonctionnement, utilisez le script d'initialisation `/usr/sbin/rcapache2`. La commande `rcapache2` prend les paramètres suivants :

`status`

Vérifie si Apache est démarré.

`start`

Démarre Apache s'il n'est pas déjà en cours d'exécution.

`startssl`

Démarre Apache avec le support de SSL s'il n'est pas déjà en cours d'exécution. Pour plus d'informations sur le support de SSL, référez-vous à Section 20.6, « Setting Up a Secure Web Server with SSL » (p. 381).

`stop`

Arrête Apache en mettant fin au processus parent.

`restart`

Arrête puis redémarre Apache. Démarre le serveur web s'il n'était pas déjà en cours d'exécution.

`try-restart`

Arrête puis redémarre Apache uniquement s'il est déjà en cours d'exécution.

`reload` ou `graceful`

Arrête le serveur web en demandant à tous les processus dérivés d'Apache de d'abord terminer leurs requêtes avant l'extinction. Chaque processus mourant est remplacé par un nouveau processus, aboutissant à un « redémarrage » complet d'Apache.

ASTUCE : Redémarrage d'Apache dans des environnements de production

Pour activer les modifications dans la configuration d'Apache sans provoquer de rupture de connexion, utilisez la commande `rcapache2 reload`.

`restart-graceful`

Démarre un second serveur web qui dessert automatiquement toutes les requêtes entrantes. L'instance précédente du serveur web continue de gérer toutes les requêtes existantes pour une période de temps définie configurée avec `GracefulShutdownTimeout`.

`rcapache2 restart-graceful` est utile aussi bien lorsque vous migrez vers une nouvelle version que lorsque vous avez modifié des options de confi-

guration qui nécessitent un redémarrage. L'utilisation de cette option garantit un temps d'arrêt du serveur minimal.

`GracefulShutdownTimeout` doit être défini, sinon `restart-graceful` se traduira par un redémarrage classique. S'il est réglé sur zéro, le serveur attendra indéfiniment jusqu'à ce que toutes les requêtes restantes aient été entièrement desservies.

Un redémarrage gracieux peut échouer si l'instance originale d'Apache n'est pas capable de libérer toutes les ressources nécessaires. Dans ce cas, la commande entraînera un arrêt gracieux.

`stop-graceful`

Arrête le serveur web après une période de temps définie configurée avec `GracefulShutdownTimeout` afin de s'assurer que les requêtes existantes puissent être terminées.

`GracefulShutdownTimeout` doit être défini, sinon `stop-graceful` se traduira par un redémarrage classique. S'il est réglé sur zéro, le serveur attendra indéfiniment jusqu'à ce que toutes les requêtes restantes aient été entièrement desservies.

`configtest` ou `extreme-configtest`

Vérifie la syntaxe des fichiers de configuration sans affecter un serveur web en cours d'exécution. Étant donné que cette vérification est forcée à chaque fois que le serveur est démarré, relancé ou redémarré, il n'est généralement pas nécessaire d'exécuter le test explicitement (si une erreur de configuration est détectée, le serveur web n'est pas démarré, relancé ou redémarré). L'option `extreme-configtest` démarre le serveur web en tant qu'utilisateur `nobody` et charge la configuration, permettant de détecter des erreurs supplémentaires. Notez que bien que la configuration soit chargée, il n'est pas possible de tester la configuration SSL car les certificats SSL ne peuvent pas être lus par `nobody`.

`probe`

Probes for the necessity of a reload (checks whether the configuration has changed) and suggests the required arguments for the `rcapache2` command.

`server-status` and `full-server-status`

Dumps a short or full status screen, respectively. Requires either `lynx` or `w3m` installed as well as the module `mod_status` enabled. In addition to that,

`status` must be added to `APACHE_SERVER_FLAGS` in the file `/etc/sysconfig/apache2`.

ASTUCE : Options supplémentaires

Si vous spécifiez des options supplémentaires à `rcapache2`, ces dernières sont transmises au serveur web.

20.4 Installation, activation et configuration de modules

Le logiciel Apache est construit de façon modulaire : mis à part quelques tâches de base, toutes les fonctionnalités sont gérées par des modules. Cela a tellement progressé que même HTTP est géré par un module (`http_core`).

Les modules d'Apache peuvent être compilés dans le binaire d'Apache au moment de la compilation ou être chargés dynamiquement durant l'exécution. Reportez-vous à Section 20.4.2, « Activation et désactivation » (p. 372) pour plus de détails sur la façon de charger des modules dynamiquement.

Les modules d'Apache peuvent être divisés en quatre catégories différentes :

Modules de base

Les modules de base sont compilés dans Apache par défaut. Apache dans openSUSE n'a que `mod_so` (nécessaire pour charger d'autres modules) et `http_core` de compilés à l'intérieur. Tous les autres sont disponibles en tant qu'objets partagés : plutôt que d'être inclus dans le binaire du serveur, ils peuvent être inclus lors de l'exécution.

Modules d'extension

En général, les modules étiquetés en tant qu'extensions sont inclus dans le paquet logiciel Apache, mais ils ne sont généralement pas compilés statiquement dans le serveur. Dans openSUSE, ils sont disponibles sous forme d'objets partagés qui peuvent être chargés dans Apache lors de l'exécution.

Modules externes

Les modules étiquetés comme externes ne sont pas inclus dans la distribution officielle d'Apache. Cependant, openSUSE fournit certains d'entre eux.

Modules multi-processus (MPMs)

Les MPMs sont responsables de l'acceptation et de la gestion des requêtes vers le serveur web, représentant le cœur du logiciel de serveur web.

20.4.1 Installation d'un module

Si vous avez fait une installation par défaut comme décrit dans Section 20.1.2, « Installation » (p. 352), les modules suivants sont déjà installés : tous les modules de base et d'extension, le module multi-processus Prefork MPM, et les modules externes `mod_php5` et `mod_python`.

Vous pouvez installer des modules externes supplémentaires en lançant YaST et en choisissant *Logiciel > Installer et supprimer des logiciels*. Choisissez maintenant *Vue > Rechercher* et cherchez *apache*. Parmi les autres paquets, la liste des résultats contient tous les modules externes disponibles pour Apache.

20.4.2 Activation et désactivation

Activez ou désactivez des modules particuliers soit manuellement, soit avec YaST. Dans YaST, les modules de langage de script (PHP5, Perl, and Python) doivent être activés ou désactivés avec la configuration du module comme décrit dans Section 20.2.3.1, « HTTP Server Wizard » (p. 362). Tous les autres modules peuvent être activés ou désactivés comme décrit dans la section intitulée « Server Modules » (p. 367).

Si vous préférez activer ou désactiver les modules manuellement, utilisez respectivement les commandes `a2enmod mod_foo` ou `a2dismod mod_foo`. `a2enmod -l` affiche une liste de tous les modules actuellement activés.

IMPORTANT : Inclusion de fichiers de configuration pour des modules externes

Si vous avez activé des modules externes manuellement, assurez-vous de charger leurs fichiers de configuration dans toutes les configurations de virtualhosts. Les fichiers de configuration pour les modules externes sont situés dans `/etc/apache2/conf.d/` et ne sont pas chargés par défaut. Si vous avez besoin des mêmes modules dans chaque virtualhost, vous pouvez inclure `*.conf` à partir de ce répertoire. Dans le cas contraire,

incluez les fichiers individuellement. Consultez `/etc/apache2/vhost.d/vhost.template` pour des exemples.

20.4.3 Modules de base et extensions

Tous les modules de base et d'extension sont décrits en détails dans la documentation Apache. Seule une brève description des principaux modules est disponible ici. Référez-vous à <http://httpd.apache.org/docs/2.2/mod/> pour plus de détails sur chaque module.

`mod_actions`

Fournit des méthodes pour exécuter un script à chaque fois qu'un certain type MIME (par exemple `application/pdf`), un fichier avec une extension spécifique (comme `.rpm`), ou un certain type de méthode (comme `GET`) est demandé. Ce module est activé par défaut.

`mod_alias`

Fournit des directives `Alias` et `Redirect` avec lesquelles vous pouvez mapper une URI vers un répertoire spécifique (`Alias`) ou rediriger une URL demandée vers un autre emplacement. Ce module est activé par défaut.

`mod_auth*`

Les modules d'authentification fournissent différentes méthodes d'authentification : l'authentification de base avec `mod_auth_basic` ou l'authentification digest avec `mod_auth_digest`. L'authentification digest est considérée comme expérimentale dans Apache 2.2.

`mod_auth_basic` and `mod_auth_digest` must be combined with an authentication provider module, `mod_authn_*` (for example, `mod_authn_file` for text file-based authentication) and with an authorization module `mod_authz_*` (for example, `mod_authz_user` for user authorization).

More information about this topic is available in the *Authentication HOWTO* at <http://httpd.apache.org/docs/2.2/howto/auth.html>.

`mod_autoindex`

Autoindex génère des listings de répertoire quand aucun fichier index n'est présent (par exemple `index.html`). L'apparence de ces listings est confi-

gurable. Ce module est activé par défaut. Cependant, les listings de répertoire sont désactivés par défaut via la directive `Options`—écrasez ce paramètre dans votre configuration de `virtualhost`. Le fichier de configuration par défaut pour ce module est situé dans `/etc/apache2/mod_autoindex-defaults.conf`.

`mod_cgi`

`mod_cgi` est nécessaire pour exécuter des scripts CGI. Ce module est activé par défaut.

`mod_deflate`

Grâce à ce module, Apache peut être configuré pour compresser à la volée des types de fichier donnés avant de les délivrer.

`mod_dir`

`mod_dir` fournit la directive `DirectoryIndex` avec laquelle vous pouvez configurer les fichiers qui sont automatiquement délivrés quand un répertoire est demandé (par défaut, `index.html`). Il fournit également une redirection automatique vers la bonne URL lorsqu'un répertoire demandé ne contient pas de slash à la fin. Ce module est activé par défaut.

`mod_env`

Contrôle l'environnement qui est transmis aux scripts CGI ou aux pages SSI. Les variables d'environnement peuvent être définies, supprimées ou transmises à partir du shell qui a lancé le processus `httpd`. Ce module est activé par défaut.

`mod_expires`

Avec `mod_expires`, vous pouvez contrôler la fréquence avec laquelle le proxy et les caches de navigateur rafraichissent vos documents en envoyant une en-tête `Expires`. Ce module est activé par défaut.

`mod_include`

`mod_include` vous permet d'utiliser Server Side Includes (SSI), qui fournit une fonctionnalité de base pour générer des pages HTML dynamiquement. Ce module est activé par défaut.

`mod_info`

Fournit une vue générale de la configuration du serveur sous `http://localhost/server-info/`. Pour des raisons de sécurité, vous devriez toujours limiter l'accès à cette URL. Par défaut, seul `localhost` est autorisé à accéder à cette URL. `mod_info` est configuré dans `/etc/apache2/mod_info.conf`.

`mod_log_config`

Avec ce module, vous pouvez configurer l'aspect des fichiers log d'Apache. Ce module est activé par défaut.

`mod_mime`

Le module mime permet de s'assurer qu'un fichier est délivré avec une en-tête MIME correcte en fonction de l'extension du nom de fichier (par exemple `text/html` pour les documents HTML). Ce module est activé par défaut.

`mod_negotiation`

Necessary for content negotiation. See <http://httpd.apache.org/docs/2.2/content-negotiation.html> for more information. This module is enabled by default.

`mod_rewrite`

Fournit la fonctionnalité de `mod_alias`, mais offre plus de fonctionnalités et de flexibilité. Avec `mod_rewrite`, vous pouvez rediriger les URL en fonction de règles multiples, d'en-têtes de requête, et bien plus.

`mod_setenvif`

Sets environment variables based on details of the client's request, such as the browser string the client sends, or the client's IP address. This module is enabled by default.

`mod_speling`

`mod_speling` attempts to automatically correct typographical errors in URLs, such as capitalization errors.

`mod_ssl`

Enables encrypted connections between Web server and clients. See Section 20.6, « Setting Up a Secure Web Server with SSL » (p. 381) for details. This module is enabled by default.

`mod_status`

Provides information on server activity and performance under `http://localhost/server-status/`. For security reasons, you should always limit access to this URL. By default, only `localhost` is allowed to access this URL. `mod_status` is configured at `/etc/apache2/mod_status.conf`

`mod_suexec`

`mod_suexec` lets you run CGI scripts under a different user and group. This module is enabled by default.

`mod_userdir`

Enables user-specific directories available under `~user/`. The `UserDir` directive must be specified in the configuration. This module is enabled by default.

20.4.4 Modules multi-processus

openSUSE provides two different multiprocessing modules (MPMs) for use with Apache:

- Prefork MPM (p. 376)
- Section 20.4.4.2, « Worker MPM » (p. 376)

20.4.4.1 Prefork MPM

The prefork MPM implements a nonthreaded, preforking Web server. It makes the Web server behave similarly to Apache version 1.x. In this version it isolates each request and handles it by forking a separate child process. Thus problematic requests cannot affect others, avoiding a lockup of the Web server.

While providing stability with this process-based approach, the prefork MPM consumes more system resources than its counterpart, the worker MPM. The prefork MPM is considered the default MPM for Unix-based operating systems.

IMPORTANT : MPMs in This Document

This document assumes Apache is used with the prefork MPM.

20.4.4.2 Worker MPM

The worker MPM provides a multi-threaded Web server. A thread is a « lighter » form of a process. The advantage of a thread over a process is its lower resource consumption. Instead of only forking child processes, the worker MPM serves requests by using threads with server processes. The preforked child processes are multi-threaded. This approach makes Apache perform better by consuming fewer system resources than the prefork MPM.

One major disadvantage is the stability of the worker MPM: if a thread becomes corrupt, all threads of a process can be affected. In the worst case, this may result in a server crash. Especially when using the Common Gateway Interface (CGI) with

Apache under heavy load, internal server errors might occur due to threads being unable to communicate with system resources. Another argument against using the worker MPM with Apache is that not all available Apache modules are thread-safe and thus cannot be used in conjunction with the worker MPM.

AVERTISSEMENT : Using PHP Modules with MPMs

Not all available PHP modules are thread-safe. Using the worker MPM with `mod_php` is strongly discouraged.

20.4.5 Modules externes

Find a list of all external modules shipped with openSUSE here. Find the module's documentation in the listed directory.

`mod-apparmor`

Adds support to Apache to provide AppArmor confinement to individual CGI scripts handled by modules like `mod_php5` and `mod_perl`.

Package Name: `apache2-mod_apparmor`

Pour plus d'informations : Partie « Confining Privileges with AppArmor » (↑Security Guide)

`mod_mono`

Using `mod_mono` allows you to run ASP.NET pages in your server.

Package Name: `apache2-mod_mono`

Configuration File: `/etc/apache2/conf.d/mod_mono.conf`

`mod_perl`

`mod_perl` enables you to run Perl scripts in an embedded interpreter. The persistent interpreter embedded in the server avoids the overhead of starting an external interpreter and the penalty of Perl start-up time.

Package Name: `apache2-mod_perl`

Configuration File: `/etc/apache2/conf.d/mod_perl.conf`

Pour plus d'informations : `/usr/share/doc/packages/apache2-mod_perl`

`mod_php5`

PHP is a server-side, cross-platform HTML embedded scripting language.

Package Name: `apache2-mod_php5`
Configuration File: `/etc/apache2/conf.d/php5.conf`
Pour plus d'informations : `/usr/share/doc/packages/apache2-mod_php5`

`mod_python`

`mod_python` allows embedding Python within the Apache HTTP server for a considerable boost in performance and added flexibility in designing Web-based applications.

Package Name: `apache2-mod_python`
Pour plus d'informations : `/usr/share/doc/packages/apache2-mod_python`

`mod_tidy`

`mod_tidy` validates each outgoing HTML page by means of the TidyLib. In case of a validation error, a page with an error list is delivered. Otherwise the original HTML page is delivered.

Package Name: `apache2-mod_tidy`
Configuration File: `/etc/apache2/mod_tidy.conf`
Pour plus d'informations : `/usr/share/doc/packages/apache2-mod_tidy`

20.4.6 Compilation

Apache can be extended by advanced users by writing custom modules. To develop modules for Apache or compile third-party modules, the package `apache2-devel` is required along with the corresponding development tools. `apache2-devel` also contains the `apxs2` tools, which are necessary for compiling additional modules for Apache.

`apxs2` enables the compilation and installation of modules from source code (including the required changes to the configuration files), which creates *dynamic shared objects* (DSOs) that can be loaded into Apache at runtime.

The `apxs2` binaries are located under `/usr/sbin`:

- `/usr/sbin/apxs2`—suitable for building an extension module that works with any MPM. The installation location is `/usr/lib/apache2`.

- `/usr/sbin/apxs2-prefork`—suitable for prefork MPM modules. The installation location is `/usr/lib/apache2-prefork`.
- `/usr/sbin/apxs2-worker`—suitable for worker MPM modules. The installation location is `/usr/lib/apache2-worker`.

Install and activate a module from source code with the following commands:

```
cd /path/to/module/source; apxs2 -cia  
mod_foo.c
```

where `-c` compiles the module, `-i` installs it, and `-a` activates it. Other options of `apxs2` are described in the `apxs2(1)` man page.

20.5 Getting CGI Scripts to Work

Apache's Common Gateway Interface (CGI) lets you create dynamic content with programs or scripts usually referred to as CGI scripts. CGI scripts can be written in any programming language. Usually, script languages such as Perl or PHP are used.

To enable Apache to deliver content created by CGI scripts, `mod_cgi` needs to be activated. `mod_alias` is also needed. Both modules are enabled by default. Refer to Section 20.4.2, « Activation et désactivation » (p. 372) for details on activating modules.

AVERTISSEMENT : CGI Security

Allowing the server to execute CGI scripts is a potential security hole. Refer to Section 20.7, « Avoiding Security Problems » (p. 388) for additional information.

20.5.1 Apache Configuration

In openSUSE, the execution of CGI scripts is only allowed in the directory `/srv/www/cgi-bin/`. This location is already configured to execute CGI scripts. If you have created a virtual host configuration (see Section 20.2.2.1, « Configuration de virtualhost » (p. 357)) and want to place your scripts in a host-specific directory, you must unlock and configure this directory.

Example 20.5 VirtualHost CGI Configuration

```
ScriptAlias /cgi-bin/ "/srv/www/www.example.com/cgi-bin/"❶

<Directory "/srv/www/www.example.com/cgi-bin/">
  Options +ExecCGI❷
  AddHandler cgi-script .cgi .pl❸
  Order allow,deny❹
  Allow from all
</Directory>
```

- ❶ Tells Apache to handle all files within this directory as CGI scripts.
- ❷ Enables CGI script execution
- ❸ Tells the server to treat files with the extensions .pl and .cgi as CGI scripts. Adjust according to your needs.
- ❹ The `Order` and `Allow` directives control the default access state and the order in which `allow` and `deny` directives are evaluated. In this case « `allow` » statements are evaluated before « `deny` » statements and universal access is enabled.

20.5.2 Running an Example Script

CGI programming differs from "regular" programming in that the CGI programs and scripts must be preceded by a MIME-Type header such as `Content-type: text/html`. This header is sent to the client, so it understands what kind of content it receives. Secondly, the script's output must be something the client, usually a Web browser, understands—HTML in most cases or plain text or images, for example.

A simple test script available under `/usr/share/doc/packages/apache2/test-cgi` is part of the Apache package. It outputs the content of some environment variables as plain text. Copy this script to either `/srv/www/cgi-bin/` or the script directory of your virtual host (`/srv/www/www.example.com/cgi-bin/`) and name it `test.cgi`.

Files accessible by the Web server should be owned by the user `root`. For additional information see Section 20.7, « Avoiding Security Problems » (p. 388). Because the Web server runs with a different user, the CGI scripts must be world-executable and world-readable. Change into the CGI directory and use the command `chmod 755 test.cgi` to apply the proper permissions.

Now call `http://localhost/cgi-bin/test.cgi` or `http://www.example.com/cgi-bin/test.cgi`. You should see the « CGI/1.0 test script report ».

20.5.3 CGI Troubleshooting

If you do not see the output of the test program but an error message instead, check the following:

CGI Troubleshooting

- Have you reloaded the server after having changed the configuration? Check with `rcapache2 probe`.
- If you have configured your custom CGI directory, is it configured properly? If in doubt, try the script within the default CGI directory `/srv/www/cgi-bin/` and call it with `http://localhost/cgi-bin/test.cgi`.
- Are the file permissions correct? Change into the CGI directory and execute `ls -l test.cgi`. Its output should start with

```
-rwxr-xr-x 1 root root
```
- Make sure that the script does not contain programming errors. If you have not changed `test.cgi`, this should not be the case, but if you are using your own programs, always make sure that they do not contain programming errors.

20.6 Setting Up a Secure Web Server with SSL

Whenever sensitive data, such as credit card information, is transferred between Web server and client, it is desirable to have a secure, encrypted connection with authentication. `mod_ssl` provides strong encryption using the secure sockets layer (SSL) and transport layer security (TLS) protocols for HTTP communication between a client and the Web server. Using SSL/TSL, a private connection between Web server and client is established. Data integrity is ensured and client and server are able to authenticate each other.

For this purpose, the server sends an SSL certificate that holds information proving the server's valid identity before any request to a URL is answered. In turn, this guarantees that the server is the uniquely correct end point for the communication. Additionally, the certificate generates an encrypted connection between client and

server that can transport information without the risk of exposing sensitive, plain-text content.

`mod_ssl` does not implement the SSL/TSL protocols itself, but acts as an interface between Apache and an SSL library. In openSUSE, the OpenSSL library is used. OpenSSL is automatically installed with Apache.

The most visible effect of using `mod_ssl` with Apache is that URLs are prefixed with `https://` instead of `http://`.

ASTUCE : Example Certificate

An example certificate for a hypothetical company « Snake Oil » is available when installing the package `apache2-example-certificates`.

20.6.1 Creating an SSL Certificate

In order to use SSL/TSL with the Web server, you need to create an SSL certificate. This certificate is needed for the authorization between Web server and client, so that each party can clearly identify the other party. To ensure the integrity of the certificate, it must be signed by a party every user trusts.

There are three types of certificates you can create: a « dummy » certificate for testing purposes only, a self-signed certificate for a defined circle of users that trust you, and a certificate signed by an independent, publicly-known certificate authority (CA).

Creating a certificate is basically a two step process. First, a private key for the certificate authority is generated then the server certificate is signed with this key.

ASTUCE : For More Information

To learn more about concepts and definitions of SSL/TSL, refer to http://httpd.apache.org/docs/2.2/ssl/ssl_intro.html.

20.6.1.1 Creating a « Dummy » Certificate

Generating a dummy certificate is simple. Just call the script `/usr/bin/genssl-cert`. It creates or overwrites the files listed below. Make use of `gensslcert's`

optional switches to fine-tune the certificate. Call `/usr/bin/gensslcert -h` for more information.

- `/etc/apache2/ssl.crt/ca.crt`
- `/etc/apache2/ssl.crt/server.crt`
- `/etc/apache2/ssl.key/server.key`
- `/etc/apache2/ssl.csr/server.csr`
- `/root/.mkcert.cfg`

A copy of `ca.crt` is also placed at `/srv/www/htdocs/CA.crt` for download.

IMPORTANT : For Testing Purposes Only

A dummy certificate should never be used on a production system. Only use it for testing purposes.

20.6.1.2 Creating a Self-Signed Certificate

If you are setting up a secure Web server for an Intranet or for a defined circle of users, it might be sufficient if you sign a certificate with your own certificate authority (CA).

Creating a self-signed certificate is an interactive nine-step process. Change into the directory `/usr/share/doc/packages/apache2` and run the following command: `./mkcert.sh make --no-print-directory /usr/bin/openssl /usr/sbin/ custom`. Do not attempt to run this command from outside this directory. The program provides a series of prompts, some of which require user input.

Procédure 20.4 *Creating a Self-Signed Certificate with `mkcert.sh`*

1 Decide the signature algorithm used for certificates

Choose RSA (R, the default), because some older browsers have problems with DSA.

2 Generating RSA private key for CA (1024 bit)

No interaction needed.

3 Generating X.509 certificate signing request for CA

Create the CA's distinguished name here. This requires you to answer a few questions, such as country name or organization name. Enter valid data, because everything you enter here later shows up in the certificate. You do not need to answer every question. If one does not apply to you or you want to leave it blank, use « . ». Common name is the name of the CA itself—choose a significant name, such as *My company* CA.

IMPORTANT : Common Name of the CA

The common name of the CA must be different from the server's common name, so do not choose the fully qualified hostname in this step.

4 Generating X.509 certificate for CA signed by itself

Choose certificate version 3 (the default).

5 Generating RSA private key for SERVER (1024 bit)

No interaction needed.

6 Generating X.509 certificate signing request for SERVER

Create the distinguished name for the server key here. Questions are almost identical to the ones already answered for the CA's distinguished name. The data entered here applies to the Web server and does not necessarily need to be identical to the CA's data (for example, if the server is located elsewhere).

IMPORTANT : Selecting a Common Name

The common name you enter here must be the fully qualified hostname of your secure server (for example, www.example.com). Otherwise the browser issues a warning that the certificate does not match the server when accessing the Web server.

7 Generating X.509 certificate signed by own CA

Choose certificate version 3 (the default).

8 Encrypting RSA private key of CA with a passphrase for security

It is strongly recommended to encrypt the private key of the CA with a password, so choose Y and enter a password.

9 Encrypting RSA private key of SERVER with a passphrase for security

Encrypting the server key with a password requires you to enter this password every time you start the Web server. This makes it difficult to automatically start the server on boot or to restart the Web server. Therefore, it is common sense to say N to this question. Keep in mind that your key is unprotected when not encrypted with a password and make sure that only authorized persons have access to the key.

IMPORTANT : Encrypting the Server Key

If you choose to encrypt the server key with a password, increase the value for `APACHE_TIMEOUT` in `/etc/sysconfig/apache2`. Otherwise you do not have enough time to enter the passphrase before the attempt to start the server is stopped unsuccessfully.

The script's result page presents a list of certificates and keys it has generated. Contrary to what the script outputs, the files have not been generated in the local directory `conf`, but to the correct locations under `/etc/apache2/`.

The last step is to copy the CA certificate file from `/etc/apache2/ssl.crt/ca.crt` to a location where your users can access it in order to incorporate it into the list of known and trusted CAs in their Web browsers. Otherwise a browser complains that the certificate was issued by an unknown authority. The certificate is valid for one year.

IMPORTANT : Self-Signed Certificates

Only use a self-signed certificate on a Web server that is accessed by people who know and trust you as a certificate authority. It is not recommended to use such a certificate for a public shop, for example.

20.6.1.3 Getting an Officially Signed Certificate

There are a number of official certificate authorities that sign your certificates. The certificate is signed by a trustworthy third party, so can be fully trusted. Publicly operating secure Web servers usually have got an officially signed certificate.

The best-known official CAs are Thawte (<http://www.thawte.com/>) or Verisign (<http://www.verisign.com>). These and other CAs are already compiled into all browsers, so certificates signed by these certificate authorities are automatically accepted by the browser.

When requesting an officially signed certificate, you do not send a certificate to the CA. Instead, issue a Certificate Signing Request (CSR). To create a CSR, call the script `/usr/share/ssl/misc/CA.sh -newreq`.

First the script asks for a password with which the CSR should be encrypted. Then you are asked to enter a distinguished name. This requires you to answer a few questions, such as country name or organization name. Enter valid data—everything you enter here later shows up in the certificate and is checked. You do not need to answer every question. If one does not apply to you or you want to leave it blank, use « . ». Common name is the name of the CA itself—choose a significant name, such as *My company* CA. Last, a challenge password and an alternative company name must be entered.

Find the CSR in the directory from which you called the script. The file is named `newreq.pem`.

20.6.2 Configuring Apache with SSL

The default port for SSL and TLS requests on the Web server side is 443. There is no conflict between a « regular » Apache listening on port 80 and an SSL/TLS-enabled Apache listening on port 443. In fact, HTTP and HTTPS can be run with the same Apache instance. Usually separate virtual hosts are used to dispatch requests to port 80 and port 443 to separate virtual servers.

IMPORTANT : Firewall Configuration

Do not forget to open the firewall for SSL-enabled Apache on port 443. This can be done with YaST as described in section « Configuring the Firewall with YaST » (Chapitre 13, *Masquerading and Firewalls*, ↑Security Guide).

The SSL module is enabled by default in the global server configuration. In case it has been disabled on your host, activate it with the following command: `a2enmod ssl`. To finally enable SSL, the server needs to be started with the flag « SSL ». To do so, call `a2enflag SSL`. If you have chosen to encrypt your server certificate with a password, you should also increase the value for `APACHE_TIMEOUT` in `/etc/sysconfig/apache2`, so you have enough time to enter the passphrase when Apache starts. Restart the server to make these changes active. A reload is not sufficient.

The virtual host configuration directory contains a template `/etc/apache2/vhosts.d/vhost-ssl.template` with SSL-specific directives that are extensively documented. Refer to Section 20.2.2.1, « Configuration de virtualhost » (p. 357) for the general virtual host configuration.

To get started, copy the template to `/etc/apache2/vhosts.d/mySSL-host.conf` and edit it. Adjusting the values for the following directives should be sufficient:

- `DocumentRoot`
- `ServerName`
- `ServerAdmin`
- `ErrorLog`
- `TransferLog`

20.6.2.1 Name-Based Virtual Hosts and SSL

By default it is not possible to run multiple SSL-enabled virtual hosts on a server with only one IP address. Name-based virtual hosting requires that Apache knows which server name has been requested. The problem with SSL connections is, that such a request can only be read after the SSL connection has already been established (by using the default virtual host). As a result, users will receive a warning message stating that the certificate does not match the server name.

openSUSE comes with an extension to the SSL protocol called Server Name Indication (SNI) addresses this issue by sending the name of the virtual domain as part of the SSL negotiation. This enables the server to « switch » to the correct virtual domain early and present the browser the correct certificate.

SNI is enabled by default on openSUSE. In order to enable Name-Based Virtual Hosts for SSL, configure the server as described in la section intitulée « Virtualhosts basés sur le nom » (p. 359) (note that you need to use port 443 rather than port 80 with SSL).

IMPORTANT : SNI Browser Support

SNI must also be supported on the client side. Although SNI is supported by most browsers, some browsers for mobile hardware as well as Internet Explorer and Safari on Windows* XP lack SNI support. See http://en.wikipedia.org/wiki/Server_Name_Indication for details.

Configure how to handle non-SNI capable browser with the directive `SSLStictSNIVHostCheck`. When set to `on` in the server configuration, non-SNI capable browser will be rejected for all virtual hosts. When set to `on` within a `VirtualHost` directive, access to this particular Host will be rejected.

When set to `off` in the server configuration, the server will behave as if not having SNI support. SSL requests will be handled by the *first* Virtual host defined (for port 443).

20.7 Avoiding Security Problems

A Web server exposed to the public Internet requires an ongoing administrative effort. It is inevitable that security issues appear, both related to the software and to accidental misconfiguration. Here are some tips for how to deal with them.

20.7.1 Up-to-Date Software

If there are vulnerabilities found in the Apache software, a security advisory will be issued by SUSE. It contains instructions for fixing the vulnerabilities, which in turn should be applied as soon as possible. The SUSE security announcements are available from the following locations:

- **Web Page** <http://www.novell.com/linux/security/securitysupport.html>

- **Mailing List Archive** <http://lists.opensuse.org/opensuse-security-announce/>
- **RSS Feed** http://www.novell.com/linux/security/suse_security.xml

20.7.2 DocumentRoot Permissions

By default in openSUSE, the `DocumentRoot` directory `/srv/www/htdocs` and the CGI directory `/srv/www/cgi-bin` belong to the user and group `root`. You should not change these permissions. If the directories are writable for all, any user can place files into them. These files might then be executed by Apache with the permissions of `wwwrun`, which may give the user unintended access to file system resources. Use subdirectories of `/srv/www` to place the `DocumentRoot` and CGI directories for your virtual hosts and make sure that directories and files belong to user and group `root`.

20.7.3 File System Access

By default, access to the whole file system is denied in `/etc/apache2/httpd.conf`. You should never overwrite these directives, but specifically enable access to all directories Apache should be able to read. For details, see la section intitulée « Configuration de virtualhost basique » (p. 361). In doing so, ensure that no critical files, such as password or system configuration files, can be read from the outside.

20.7.4 CGI Scripts

Interactive scripts in Perl, PHP, SSI, or any other programming language can essentially run arbitrary commands and therefore present a general security issue. Scripts that will be executed from the server should only be installed from sources the server administrator trusts—allowing users to run their own scripts is generally not a good idea. It is also recommended to do security audits for all scripts.

To make the administration of scripts as easy as possible, it is common practice to limit the execution of CGI scripts to specific directories instead of globally allowing them. The directives `ScriptAlias` and `Option ExecCGI` are used for configuration. The openSUSE default configuration does not allow execution of CGI scripts from everywhere.

All CGI scripts run as the same user, so different scripts can potentially conflict with each other. The module suEXEC lets you run CGI scripts under a different user and group.

20.7.5 User Directories

When enabling user directories (with `mod_userdir` or `mod_rewrite`) you should strongly consider not allowing `.htaccess` files, which would allow users to overwrite security settings. At least you should limit the user's engagement by using the directive `AllowOverride`. In openSUSE, `.htaccess` files are enabled by default, but the user is not allowed to overwrite any `Option` directives when using `mod_userdir` (see the `/etc/apache2/mod_userdir.conf` configuration file).

20.8 Troubleshooting

If Apache does not start, the Web page is not accessible, or users cannot connect to the Web server, it is important to find the cause of the problem. Here are some typical places to look for error explanations and important things to check:

Output of `rcapache2`

Instead of starting and stopping the Web server with the binary `/usr/sbin/httpd2`, rather use the `rcapache2` script instead (described in Section 20.3, « Démarrage et arrêt d'Apache » (p. 368)). It is verbose about errors, and it even provides tips and hints for fixing configuration errors.

Log Files and Verbosity

In case of both fatal and nonfatal errors, check the Apache log files for causes, mainly the error log file located at `/var/log/apache2/error_log` by default. Additionally, you can control the verbosity of the logged messages with the `LogLevel` directive if more detail is needed in the log files.

ASTUCE : A Simple Test

Watch the Apache log messages with the command `tail -F /var/log/apache2/my_error_log`. Then run `rcapache2 restart`. Now, try to connect with a browser and check the output.

Firewall and Ports

A common mistake is to not open the ports for Apache in the firewall configuration of the server. If you configure Apache with YaST, there is a separate option available to take care of this specific issue (see Section 20.2.3, « Configuring Apache with YaST » (p. 362)). If you are configuring Apache manually, open firewall ports for HTTP and HTTPS via YaST's firewall module.

If the error cannot be tracked down with the help of any these, check the online Apache bug database at http://httpd.apache.org/bug_report.html. Additionally, the Apache user community can be reached via a mailing list available at <http://httpd.apache.org/userslist.html>. A recommended news-group is comp.infosystems.www.servers.unix.

20.9 For More Information

The package `apache2-doc` contains the complete Apache manual in various localizations for local installation and reference. It is not installed by default—the quickest way to install it is to use the command `zypper in apache2-doc`. Once installed, the Apache manual is available at <http://localhost/manual/>. You may also access it on the Web at <http://httpd.apache.org/docs-2.2/>. SUSE-specific configuration hints are available in the directory `/usr/share/doc/packages/apache2/README.*`.

20.9.1 Apache 2.2

For a list of new features in Apache 2.2, refer to http://httpd.apache.org/docs/2.2/new_features_2_2.html. Information about upgrading from version 2.0 to 2.2 is available at <http://httpd.apache.org/docs-2.2/upgrading.html>.

20.9.2 Apache Modules

More information about external Apache modules that are briefly described in Section 20.4.5, « Modules externes » (p. 377) is available at the following locations:

`mod-apparmor`

<http://en.opensuse.org/SDB:AppArmor>

mod_mono
http://www.mono-project.com/Mod_mono

mod_perl
<http://perl.apache.org/>

mod_php5
<http://www.php.net/manual/en/install.unix.apache2.php>

mod_python
<http://www.modpython.org/>

mod_tidy
<http://mod-tidy.sourceforge.net/>

20.9.3 Développement

More information about developing Apache modules or about getting involved in the Apache Web server project are available at the following locations:

Apache Developer Information

<http://httpd.apache.org/dev/>

Apache Developer Documentation

<http://httpd.apache.org/docs/2.2/developer/>

Writing Apache Modules with Perl and C

<http://www.modperl.com/>

20.9.4 Miscellaneous Sources

If you experience difficulties specific to Apache in openSUSE, take a look at the openSUSE wiki at <http://old-en.opensuse.org/Apache>. The history of Apache is provided at http://httpd.apache.org/ABOUT_APACHE.html. This page also explains why the server is called Apache.

Setting up an FTP server with YaST

Using the YaST *FTP Server* module, you can configure your machine to function as an FTP (File Transfer Protocol) server. Anonymous and/or authenticated users can connect to your machine and download files using the FTP protocol. Depending on the configuration, they can also upload files to the FTP server. YaST provides a unified configuration interface for various FTP server daemons installed on your system.

You can use the YaST *FTP Server* configuration module to configure two different FTP server daemons:

- `vsftpd` (Very Secure FTP Daemon) and
- `pure-ftpd`

Only installed servers can be configured. Standard media do not contain the `pure-ftpd` package. However, if the `pure-ftpd` package is installed from another repository, it can be configured using the module. The `vsftpd` and `pure-ftpd` servers have slightly different configuration options, especially in the Experts Settings dialog. This chapter describes the settings of the `vsftpd` server, being the default server for . If the FTP Server module is not available in your system, install the `yast2-ftp-server` package. To configure the FTP server using , follow these steps: Open Control Center and choose Network Services FTP Server or run the `yast2 ftp-server` command as . If there is not any FTP server installed in your system, you will be asked which server to install when the FTP Server module starts. Choose a server (`vsftpd` is the standard server for) and confirm the dialog. If there are two servers installed, choose the preferred server and click OK. In the Start-Up dialog, configure the options for

starting of the FTP server. For more information, see . In the General dialog, configure FTP directories, welcome message, file creation masks and various other parameters. For more information, see . In the Performance dialog, set the parameters that affect the load on the FTP server. For more information, see . In the Authentication dialog, set whether the FTP server should be available for anonymous and/or authenticated users. For more information, see . In the Expert Settings dialog, configure the operation mode of the FTP server, SSL connections and firewall settings. For more information, see . Press Finish to save the configurations. Starting the FTP server In the Service Start frame of the FTP Start-Up dialog set the way the FTP server is started up. You can choose between starting the server automatically during the system boot and starting it manually. If the FTP server should be started only after an FTP connection request, choose Via xinetd. The current status of the FTP server is shown in the Switch On and Off frame of the FTP Start-Up dialog. Start the FTP server by clicking Start FTP Now. To stop the server, click Stop FTP Now. After having changed the settings of the server click Save Settings and Restart FTP Now. Your configurations will be saved by leaving the configuration module with Finish. The Selected Service frame of the FTP Start-Up dialog shows which FTP server is used: either vsftpd or pure-ftpd. If both servers are installed, you can switch between them—the current configuration will automatically be converted. The pure-ftpd package is not included in the standard media so you have to install it from a different installation source if you want to use it. FTP Server Configuration FTP General Settings In the General Settings frame of the FTP General Settings dialog you can set the Welcome message which is shown after connecting to the FTP server. If you check the Chroot Everyone option, all local users will be placed in a chroot jail in their home directory after login. This option has security implications, especially if the users have upload permission or shell access, so be careful enabling this option. If you check the Verbose Logging option, all FTP requests and responses are logged. You can limit permissions of files created by anonymous and/or authenticated users with umask. Set the file creation mask for anonymous users in Umask for Anonymous and the file creation mask for authenticated users in Umask for Authenticated Users. The masks should be entered as octal numbers with a leading zero. For more information about umask, see the umask man page (man 1p umask). In the FTP Directories frame set the directories used for anonymous and authorized users. With Browse, you can select a directory to be used from the local filesystem. The default FTP directory for anonymous users is /srv/ftp. Note that vsftpd does not allow this directory to be writable for all users. The subdirectory upload with write permissions for anonymous users is created instead. Write Permissions in FTP Directory The pure-ftpd server allows the FTP directory for anonymous users to be writable. When switching between servers, make sure you remove the write permissions in the directory that was used with pure-ftpd before switching back to the vsftpd server. FTP

Performance Settings In the Performance dialog set the parameters which affect the load on the FTP server. Max Idle Time is the maximum time (in minutes) the remote client may spend between FTP commands. In case of longer inactivity, the remote client is disconnected. Max Clients for One IP determines the maximum number of clients which can be connected from a single IP address. Max Clients determines the maximum number of clients which may be connected. Any additional clients will get an error message. The maximum data transfer rate (in KB/s) is set in Local Max Rate for local authenticated users, and in Anonymous Max Rate for anonymous clients respectively. The default value for the rate settings is 0, which means unlimited data transfer rate. **Authentication** In the Enable/Disable Anonymous and Local Users frame of the Authentication dialog, you are able to set which users are allowed to access your FTP server. You can choose between the following options: granting access to anonymous users only, to authenticated users only (with accounts on the system) or to both types of users. If you want to allow users to upload files to the FTP server, check Enable Upload in the Uploading frame of the Authentication dialog. Here you are able to allow uploading or creating directories even for anonymous users by checking the respective box. **vsftp—Allowing File Upload for Anonymous Users** If a vsftpd server is used and you want anonymous users to be able to upload files or create directories, a subdirectory with writing permissions for all users has to be created in the anonymous FTP directory. **Expert Settings** An FTP server can run in active or in passive mode. By default the server runs in passive mode. To switch into active mode, just uncheck Enable Passive Mode option in Expert Settings dialog. You can also change the range of ports on the server used for the data stream by tweaking the Min Port for Pas. Mode and Max Port for Pas. Mode options. If you want encrypted communication between clients and the server, you can Enable SSL. Check the versions of the protocol to be supported and specify the DSA certificate to be used for SSL encrypted connections. If your system is protected by a firewall, check Open Port in Firewall to enable a connection to the FTP server. For more information For more information about FTP servers read the manual pages of pure-ftpd, vsftpd and vsftpd.conf.

Partie V. Mobilité

Mobile Computing with Linux

22

Mobile computing is mostly associated with laptops, PDAs and cellular phones (and the data exchange between them). Mobile hardware components, such as external hard disks, flash drives, or digital cameras, can be connected to laptops or desktop systems. A number of software components are involved in mobile computing scenarios and some applications are tailor-made for mobile use.

22.1 Laptops

The hardware of laptops differs from that of a normal desktop system. This is because criteria like exchangeability, space requirements and power consumption must be taken into account. The manufacturers of mobile hardware have developed standard interfaces like PCMCIA (Personal Computer Memory Card International Association), Mini PCI and Mini PCIe that can be used to extend the hardware of laptops. The standards cover memory cards, network interface cards, ISDN (and modem cards) and external hard disks.

ASTUCE : openSUSE and Tablet PCs

openSUSE also supports Tablet PCs. Tablet PCs come with a touchpad/digitizer that allows you to use a digital pen or even fingertips to edit data right on the screen instead of using mouse and keyboard. They are installed and configured much like any other system. For a detailed introduction to the installation and configuration of Tablet PCs, refer to Chapitre 26, *Using Tablet PCs* (p. 457).

22.1.1 Power Conservation

The inclusion of energy-optimized system components during laptop manufacturing contributes to their suitability for use without access to the electrical power grid. Their contribution towards conservation of power is at least as important as that of the operating system. openSUSE® supports various methods that influence the power consumption of a laptop and have varying effects on the operating time under battery power. The following list is in descending order of contribution towards power conservation:

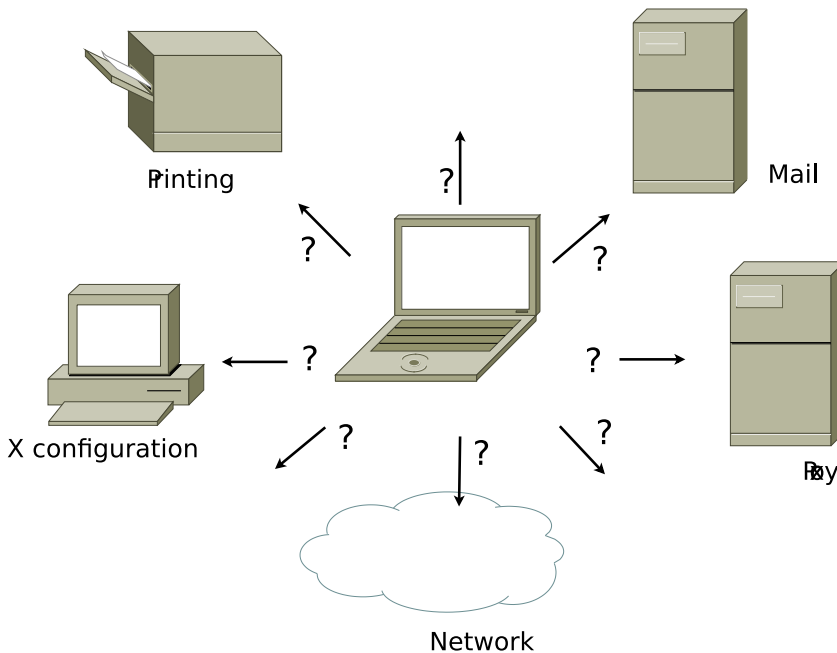
- Throttling the CPU speed.
- Switching off the display illumination during pauses.
- Manually adjusting the display illumination.
- Disconnecting unused, hotplug-enabled accessories (USB CD-ROM, external mouse, unused PCMCIA cards, WLAN, etc.).
- Spinning down the hard disk when idling.

Detailed background information about power management in openSUSE is provided in Chapitre 23, *Power Management* (p. 409).

22.1.2 Integration in Changing Operating Environments

Your system needs to adapt to changing operating environments when used for mobile computing. Many services depend on the environment and the underlying clients must be reconfigured. openSUSE handles this task for you.

Figure 22.1 *Integrating a Mobile Computer in an Existing Environment*



The services affected in the case of a laptop commuting back and forth between a small home network and an office network are:

Network

This includes IP address assignment, name resolution, Internet connectivity and connectivity to other networks.

Printing

A current database of available printers and an available print server must be present, depending on the network.

E-Mail and Proxies

As with printing, the list of the corresponding servers must be current.

X (Graphical Environment)

If your laptop is temporarily connected to a projector or an external monitor, different display configurations must be available.

openSUSE offers several ways of integrating laptops into existing operating environments:

NetworkManager

NetworkManager is especially tailored for mobile networking on laptops. It provides a means to easily and automatically switch between network environments or different types of networks such as mobile broadband (such as GPRS, EDGE, or 3G), wireless LAN, and Ethernet. NetworkManager supports WEP and WPA-PSK encryption in wireless LANs. It also supports dial-up connections (with smpppd). Both desktop environments (GNOME and KDE) include a front-end for NetworkManager. For more information about the desktop applets, see Section 25.4, « Using the KDE NetworkManager Front-End » (p. 442) and Section 25.5, « Using GNOME NetworkManager » (p. 446).

Tableau 22.1 *Use Cases for NetworkManager*

My computer...	Use NetworkManager
is a laptop	Yes
is sometimes attached to different networks	Yes
provides network services (such as DNS or DHCP)	No
only uses a static IP address	No

Use the YaST tools to configure networking whenever NetworkManager should not handle network configuration.

ASTUCE : DNS configuration and various types of network connections

If you travel frequently with your laptop and change different types of network connections, NetworkManager works fine when all DNS addresses are assigned correctly assigned with DHCP. If some of your connections use static DNS address(es), add it to the `NETCONFIG_DNS_STATIC_SERVERS` option in `/etc/sysconfig/network/config`.

SLP

The service location protocol (SLP) simplifies the connection of a laptop to an existing network. Without SLP, the administrator of a laptop usually requires detailed knowledge of the services available in a network. SLP broadcasts the availability of a certain type of service to all clients in a local network. Applications that support SLP can process the information dispatched by SLP and be configured automatically. SLP can also be used to install a system, minimizing the effort of searching for a suitable installation source. Find detailed information about SLP in Chapitre 14, *SLP Services in the Network* (p. 279).

22.1.3 Software Options

There are various special task areas in mobile use that are covered by dedicated software: system monitoring (especially the battery charge), data synchronization, and wireless communication with peripherals and the Internet. The following sections cover the most important applications that openSUSE provides for each task.

22.1.3.1 System Monitoring

Two KDE system monitoring tools are provided by openSUSE:

Power Management

Power Management is an application which lets you adjust energy saving related behavior of the KDE desktop. You can typically access it via the *Battery Monitor* tray icon, which changes according to the type of the current power supply. Other way to open its configuration dialog is through the *Kickoff Application Launcher: Applications > Configure Desktop > Advanced > Power Management*.

Click the *Battery Monitor* tray icon to access options to configure its behavior. You can choose one of five displayed power profiles which best fits your needs. For example, the *Presentation* scheme disables the screen saver and the power management in general, so that your presentation is not interrupted by system events. Click *More...* to open a more complex configuration screen. Here you can edit individual profiles and set advanced power management options and notifications, such as what to do when the laptop lid has been closed, or when the battery charge is low.

System Monitor

System Monitor (also called *KSysguard*) gathers measurable system parameters into one monitoring environment. It presents the output information in 2 tabs

by default. *Process Table* gives detailed information about currently running processes, such as CPU load, memory usage, or process ID number and nice value. The presentation and filtering of the collected data can be customized — to add a new type of process information, left-click on the process table header and choose which column to hide or add to the view. It is also possible to monitor different system parameters in various data pages or collect the data of various machines in parallel over the network. KSysguard can also run as a daemon on machines without a KDE environment. Find more information about this program in its integrated help function or in the SUSE help pages.

In the GNOME environment use *Power Management Preferences* and *System Monitor*.

22.1.3.2 Synchronizing Data

When switching between working on a mobile machine disconnected from the network and working at a networked workstation in an office, it is necessary to keep processed data synchronized across all instances. This could include e-mail folders, directories and individual files that need to be present for work on the road as well as at the office. The solution in both cases is as follows:

Synchronizing E-Mail

Use an IMAP account for storing your e-mails in the office network. Then access the e-mails from the workstation using any disconnected IMAP-enabled e-mail client, like Mozilla Thunderbird Mail, Evolution, or KMail. The e-mail client must be configured so that the same folder is always accessed for *Sent* messages. This ensures that all messages are available along with their status information after the synchronization process has completed. Use an SMTP server implemented in the mail client for sending messages instead of the system-wide MTA postfix or sendmail to receive reliable feedback about unsent mail.

Synchronizing Files and Directories

There are several utilities suitable for synchronizing data between a laptop and a workstation. One of the most widely used is a command-line tool called `rsync`. For more information, see Chapitre 27, *Copying and Sharing Files* (p. 467).

22.1.3.3 Wireless Communication

As well as connecting to a home or office network with a cable, a laptop can also use wireless connection to access other computers, peripherals, cellular phones or PDAs. Linux supports three types of wireless communication:

WLAN

With the largest range of these wireless technologies, WLAN is the only one suitable for the operation of large and sometimes even spatially separate networks. Single machines can connect with each other to form an independent wireless network or access the Internet. Devices called *access points* act as base stations for WLAN-enabled devices and act as intermediaries for access to the Internet. A mobile user can switch among access points depending on location and which access point is offering the best connection. Like in cellular telephony, a large network is available to WLAN users without binding them to a specific location for accessing it. Find details about WLAN in Chapitre 24, *Wireless LAN* (p. 419).

Bluetooth

Bluetooth has the broadest application spectrum of all wireless technologies. It can be used for communication between computers (laptops) and PDAs or cellular phones, as can IrDA. It can also be used to connect various computers within range. Bluetooth is also used to connect wireless system components, like a keyboard or a mouse. The range of this technology is, however, not sufficient to connect remote systems to a network. WLAN is the technology of choice for communicating through physical obstacles like walls.

IrDA

IrDA is the wireless technology with the shortest range. Both communication parties must be within viewing distance of each other. Obstacles like walls cannot be overcome. One possible application of IrDA is the transmission of a file from a laptop to a cellular phone. The short path from the laptop to the cellular phone is then covered using IrDA. The long range transport of the file to the recipient of the file is handled by the mobile network. Another application of IrDA is the wireless transmission of printing jobs in the office.

22.1.4 Data Security

Ideally, you protect data on your laptop against unauthorized access in multiple ways. Possible security measures can be taken in the following areas:

Protection against Theft

Always physically secure your system against theft whenever possible. Various securing tools (like chains) are available in retail stores.

Strong Authentication

Use biometric authentication in addition to standard authentication via login and password. openSUSE supports fingerprint authentication. For more details, see Chapitre 7, *Using the Fingerprint Reader* (↑Security Guide).

Securing Data on the System

Important data should not only be encrypted during transmission, but also on the hard disk. This ensures its safety in case of theft. The creation of an encrypted partition with openSUSE is described in Chapitre 10, *Encrypting Partitions and Files* (↑Security Guide). Another possibility is to create encrypted home directories when adding the user with YaST.

IMPORTANT : Data Security and Suspend to Disk

Encrypted partitions are not unmounted during a suspend to disk event. Thus, all data on these partitions is available to any party who manages to steal the hardware and issue a resume of the hard disk.

Network Security

Any transfer of data should be secured, no matter how the transfer is done. Find general security issues regarding Linux and networks in Chapitre 1, *Security and Confidentiality* (↑Security Guide). Security measures related to wireless networking are provided in Chapitre 24, *Wireless LAN* (p. 419).

22.2 Mobile Hardware

openSUSE supports the automatic detection of mobile storage devices over FireWire (IEEE 1394) or USB. The term *mobile storage device* applies to any kind of FireWire or USB hard disk, USB flash drive, or digital camera. These devices are automatically detected and configured as soon as they are connected with the system over the corresponding interface. The file managers of both GNOME and KDE offer flexible handling of mobile hardware items. To unmount any of these media safely, use the *Safely Remove* (KDE) or *Unmount* (GNOME) feature of either file manager.

External Hard Disks (USB and FireWire)

As soon as an external hard disk is correctly recognized by the system, its icon appears in the file manager. Clicking the icon displays the contents of the drive. It is possible to create folders and files here and edit or delete them. To rename a hard disk from the name it had been given by the system, select the correspon-

ding menu item from the menu that opens when the icon is right-clicked. This name change is limited to display in the file manager. The descriptor by which the device is mounted in `/media` remains unaffected by this.

USB Flash Drives

These devices are handled by the system just like external hard disks. It is similarly possible to rename the entries in the file manager.

Digital Cameras (USB and FireWire)

Digital cameras recognized by the system also appear as external drives in the overview of the file manager. KDE allows reading and accessing the pictures at the URL `camera:/`. The images can then be processed using `digiKam` or `f-spot`. For advanced photo processing, use `GIMP`.

22.3 Cellular Phones and PDAs

A desktop system or a laptop can communicate with a cellular phone via Bluetooth or IrDA. Some models support both protocols and some only one of the two. The usage areas for the two protocols and the corresponding extended documentation has already been mentioned in Section 22.1.3.3, « Wireless Communication » (p. 404). The configuration of these protocols on the cellular phones themselves is described in their manuals.

The support for synchronizing with handheld devices manufactured by Palm, Inc., is already built into `Evolution` and `Kontact`. Initial connection with the device is easily performed with the assistance of a wizard. Once the support for Palm Pilots is configured, it is necessary to determine which type of data should be synchronized (addresses, appointments, etc.).

A more sophisticated synchronization solution is available with the program `open-sync` (see the package `libopendsync` and the respective plug-ins for the various devices).

22.4 For More Information

The central point of reference for all questions regarding mobile devices and Linux is <http://tuxmobil.org/>. Various sections of that Web site deal with the hard-

ware and software aspects of laptops, PDAs, cellular phones and other mobile hardware.

A similar approach to that of <http://tuxmobil.org/> is made by <http://www.linux-on-laptops.com/>. Information about laptops and handhelds can be found here.

SUSE maintains a mailing list in German dedicated to the subject of laptops. See <http://lists.opensuse.org/opensuse-mobile-de/>. On this list, users and developers discuss all aspects of mobile computing with openSUSE. Postings in English are answered, but the majority of the archived information is only available in German. Use <http://lists.opensuse.org/opensuse-mobile/> for English postings.

Information about OpenSync is available on <http://en.opensuse.org/OpenSync>.

Power Management

Power management is especially important on laptop computers, but is also useful on other systems. ACPI (Advanced Configuration and Power Interface) is available on all modern computers (laptops, desktops, and servers). Power management technologies require suitable hardware and BIOS routines. Most laptops and many modern desktops and servers meet these requirements. It is also possible to control CPU frequency scaling to save power or decrease noise.

23.1 Power Saving Functions

Power saving functions are not only significant for the mobile use of laptops, but also for desktop systems. The main functions and their use in ACPI are:

Standby

not supported.

Suspend (to memory)

This mode writes the entire system state to the RAM. Subsequently, the entire system except the RAM is put to sleep. In this state, the computer consumes very little power. The advantage of this state is the possibility of resuming work at the same point within a few seconds without having to boot and restart applications. This function corresponds to the ACPI state S3. The support of this state is still under development and therefore largely depends on the hardware.

Hibernation (suspend to disk)

In this operating mode, the entire system state is written to the hard disk and the system is powered off. There must be a swap partition at least as big as the RAM to write all the active data. Reactivation from this state takes about 30 to 90 seconds. The state prior to the suspend is restored. Some manufacturers offer useful hybrid variants of this mode, such as RediSafe in IBM Thinkpads. The corresponding ACPI state is S4. In Linux, suspend to disk is performed by kernel routines that are independent from ACPI.

Battery Monitor

ACPI checks the battery charge status and provides information about it. Additionally, it coordinates actions to perform when a critical charge status is reached.

Automatic Power-Off

Following a shutdown, the computer is powered off. This is especially important when an automatic shutdown is performed shortly before the battery is empty.

Processor Speed Control

In connection with the CPU, energy can be saved in three different ways: frequency and voltage scaling (also known as PowerNow! or Speedstep), throttling and putting the processor to sleep (C-states). Depending on the operating mode of the computer, these methods can also be combined.

23.2 Advanced Configuration and Power Interface (ACPI)

ACPI was designed to enable the operating system to set up and control the individual hardware components. ACPI supersedes both Power Management Plug and Play (PnP) and Advanced Power Management (APM). It delivers information about the battery, AC adapter, temperature, fan and system events, like « close lid » or « battery low. »

The BIOS provides tables containing information about the individual components and hardware access methods. The operating system uses this information for tasks like assigning interrupts or activating and deactivating components. Because the operating system executes commands stored in the BIOS, the functionality depends on the BIOS implementation. The tables ACPI can detect and load are reported in `/var/log/boot.msg`. See Section 23.2.2, « Troubleshooting » (p. 411) for more information about troubleshooting ACPI problems.

23.2.1 Controlling the CPU Performance

The CPU can save energy in three ways:

- Frequency and Voltage Scaling
- Throttling the Clock Frequency (T-states)
- Putting the Processor to Sleep (C-states)

Depending on the operating mode of the computer, these methods can be combined. Saving energy also means that the system heats up less and the fans are activated less frequently.

Frequency scaling and throttling are only relevant if the processor is busy, because the most economic C-state is applied anyway when the processor is idle. If the CPU is busy, frequency scaling is the recommended power saving method. Often the processor only works with a partial load. In this case, it can be run with a lower frequency. Usually, dynamic frequency scaling controlled by the kernel on-demand governor is the best approach.

Throttling should be used as the last resort, for example, to extend the battery operation time despite a high system load. However, some systems do not run smoothly when they are throttled too much. Moreover, CPU throttling does not make sense if the CPU has little to do.

For in-depth information, refer to Chapitre 11, *Power Management* (↑System Analysis and Tuning Guide).

23.2.2 Troubleshooting

There are two different types of problems. On one hand, the ACPI code of the kernel may contain bugs that were not detected in time. In this case, a solution will be made available for download. More often, the problems are caused by the BIOS. Sometimes, deviations from the ACPI specification are purposely integrated in the BIOS to circumvent errors in the ACPI implementation of other widespread operating systems. Hardware components that have serious errors in the ACPI implementation

are recorded in a blacklist that prevents the Linux kernel from using ACPI for these components.

The first thing to do when problems are encountered is to update the BIOS. If the computer does not boot at all, one of the following boot parameters may be helpful:

`pci=noacpi`

Do not use ACPI for configuring the PCI devices.

`acpi=ht`

Only perform a simple resource configuration. Do not use ACPI for other purposes.

`acpi=off`

Disable ACPI.

AVERTISSEMENT : Problems Booting without ACPI

Some newer machines (especially SMP systems and AMD64 systems) need ACPI for configuring the hardware correctly. On these machines, disabling ACPI can cause problems.

Sometimes, the machine is confused by hardware that is attached over USB or Fire-Wire. If a machine refuses to boot, unplug all unneeded hardware and try again.

Monitor the boot messages of the system with the command `dmesg | grep -2i acpi` (or all messages, because the problem may not be caused by ACPI) after booting. If an error occurs while parsing an ACPI table, the most important table—the DSDT (*Differentiated System Description Table*)—can be replaced with an improved version. In this case, the faulty DSDT of the BIOS is ignored. The procedure is described in Section 23.4.1, « ACPI Activated with Hardware Support but Functions Do Not Work » (p. 415).

In the kernel configuration, there is a switch for activating ACPI debug messages. If a kernel with ACPI debugging is compiled and installed, experts searching for an error can be supported with detailed information.

If you experience BIOS or hardware problems, it is always advisable to contact the manufacturers. Especially if they do not always provide assistance for Linux, they should be confronted with the problems. Manufacturers will only take the issue seriously if they realize that an adequate number of their customers use Linux.

23.2.2.1 For More Information

- <http://tldp.org/HOWTO/ACPI-HOWTO/> (detailed ACPI HOWTO, contains DSDT patches)
- <http://www.acpi.info> (Advanced Configuration & Power Interface Specification)
- <http://www.lesswatts.org/projects/acpi/> (the ACPI4Linux project at Sourceforge)
- <http://acpi.sourceforge.net/dsdt/index.php> (DSDT patches by Bruno Ducrot)

23.3 Rest for the Hard Disk

In Linux, the hard disk can be put to sleep entirely if it is not needed or it can be run in a more economic or quieter mode. On modern laptops, you do not need to switch off the hard disks manually, because they automatically enter an economic operating mode whenever they are not needed. However, if you want to maximize power savings, test some of the following methods, using the `hdparm` command.

It can be used to modify various hard disk settings. The option `-y` instantly switches the hard disk to the standby mode. `-Y` puts it to sleep. `hdparm -S x` causes the hard disk to be spun down after a certain period of inactivity. Replace `x` as follows: 0 disables this mechanism, causing the hard disk to run continuously. Values from 1 to 240 are multiplied by 5 seconds. Values from 241 to 251 correspond to 1 to 11 times 30 minutes.

Internal power saving options of the hard disk can be controlled with the option `-B`. Select a value from 0 to 255 for maximum saving to maximum throughput. The result depends on the hard disk used and is difficult to assess. To make a hard disk quieter, use the option `-M`. Select a value from 128 to 254 for quiet to fast.

Often, it is not so easy to put the hard disk to sleep. In Linux, numerous processes write to the hard disk, waking it up repeatedly. Therefore, it is important to understand how Linux handles data that needs to be written to the hard disk. First, all data is buffered in the RAM. This buffer is monitored by the `pdflush` daemon. When

the data reaches a certain age limit or when the buffer is filled to a certain degree, the buffer content is flushed to the hard disk. The buffer size is dynamic and depends on the size of the memory and the system load. By default, `pdflush` is set to short intervals to achieve maximum data integrity. It checks the buffer every 5 seconds and writes the data to the hard disk. The following variables are interesting:

`/proc/sys/vm/dirty_writeback_centisecs`

Contains the delay until a `pdflush` thread wakes up (in hundredths of a second).

`/proc/sys/vm/dirty_expire_centisecs`

Defines after which timeframe a dirty page should be written out latest. Default is 3000, which means 30 seconds.

`/proc/sys/vm/dirty_background_ratio`

Maximum percentage of dirty pages until `pdflush` begins to write them. Default is 5%.

`/proc/sys/vm/dirty_ratio`

When the dirty page exceeds this percentage of the total memory, processes are forced to write dirty buffers during their time slice instead of continuing to write.

AVERTISSEMENT : Impairment of the Data Integrity

Changes to the `pdflush` daemon settings endanger the data integrity.

Apart from these processes, journaling file systems, like `ReiserFS`, `Ext3`, `Ext4` and others write their metadata independently from `pdflush`, which also prevents the hard disk from spinning down. To avoid this, a special kernel extension has been developed for mobile devices. To make use of the extension, install the `laptop-mode-tools` package and see `/usr/src/linux/Documentation/laptops/laptop-mode.txt` for details.

Another important factor is the way active programs behave. For example, good editors regularly write hidden backups of the currently modified file to the hard disk, causing the disk to wake up. Features like this can be disabled at the expense of data integrity.

In this connection, the mail daemon `postfix` makes use of the variable `POSTFIX_LAPTOP`. If this variable is set to `yes`, `postfix` accesses the hard disk far less frequently.

23.4 Troubleshooting

All error messages and alerts are logged in the file `/var/log/messages`. The following sections cover the most common problems.

23.4.1 ACPI Activated with Hardware Support but Functions Do Not Work

If you experience problems with ACPI, search the output of `dmesg` for ACPI-specific messages by using the command `dmesg|grep -i acpi`.

A BIOS update may be required to resolve the problem. Go to the home page of your laptop manufacturer, look for an updated BIOS version, and install it. Ask the manufacturer to comply with the latest ACPI specification. If the errors persist after the BIOS update, proceed as follows to replace the faulty DSDT table in your BIOS with an updated DSDT:

Procédure 23.1 *Updating the DSDT Table in the BIOS*

For the procedure below, make sure the following packages are installed: `kernel-source`, `acpica`, and `mkinitrd`.

- 1 Download the DSDT for your system from <http://acpi.sourceforge.net/dsdt/index.php>. Check if the file is decompressed and compiled as shown by the file extension `.aml` (ACPI machine language). If this is the case, continue with step 3.
- 2 If the file extension of the downloaded table is `.asl` (ACPI source language) instead, compile it by executing the following command:

```
iasl -sa file.asl
```
- 3 Copy the (resulting) file `DSDT.aml` to any location (`/etc/DSDT.aml` is recommended).
- 4 Edit `/etc/sysconfig/kernel` and adapt the path to the DSDT file accordingly.

- 5 Start `mkinitrd`. Whenever you install the kernel and use `mkinitrd` to create an `initrd` file, the modified DSDT is integrated and loaded when the system is booted.

23.4.2 CPU Frequency Does Not Work

Refer to the kernel sources to see if your processor is supported. You may need a special kernel module or module option to activate CPU frequency control. If the `kernel-source` package is installed, this information is available in `/usr/src/linux/Documentation/cpu-freq/*`.

23.4.3 Suspend and Standby Do Not Work

ACPI systems may have problems with suspend and standby due to a faulty DSDT implementation (BIOS). If this is the case, update the BIOS.

When the system tries to unload faulty modules, the system is arrested or the suspend event is not triggered. The same can also happen if you do not unload modules or stop services that prevent a successful suspend. In both cases, try to identify the faulty module that prevented the sleep mode. The log file `/var/log/pm-suspend.log` contains detailed information about what is going on and where possible errors are. Modify the `SUSPEND_MODULES` variable in `/usr/lib/pm-utils/defaults` to unload problematic modules prior to a suspend or standby.

Refer to <http://old-en.opensuse.org/Pm-utils> and http://en.opensuse.org/SDB:Suspend_to_RAM to get more detailed information on how to modify the suspend and resume process.

23.5 For More Information

- <http://www.acpi.info> (Advanced Configuration and Power Interface Specification)
- <http://www.lesswatts.org/projects/acpi/> (the ACPI4Linux project at Sourceforge)

- <http://acpi.sourceforge.net/dsdt/index.php> (DSDT patches by Bruno Ducrot)
- http://en.opensuse.org/SDB:Suspend_to_RAM—How to get Suspend to RAM working
- <http://old-en.opensuse.org/Pm-utils>—How to modify the general suspend framework

Wireless LAN

Wireless LANs, or Wireless Local Area Network (WLANs), have become an indispensable aspect of mobile computing. Today, most laptops have built-in WLAN cards. This chapter describes how to set up a WLAN card with YaST, encrypt transmissions, and use tips and tricks. Alternatively, you can configure and manage WLAN access with NetworkManager. For details, refer to Chapitre 25, *Using NetworkManager* (p. 437).

24.1 WLAN Standards

WLAN cards communicate using the 802.11 standard, prepared by the IEEE organization. Originally, this standard provided for a maximum transmission rate of 2 Mbit/s. Meanwhile, several supplements have been added to increase the data rate. These supplements define details such as the modulation, transmission output, and transmission rates (see Tableau 24.1, « Overview of Various WLAN Standards » (p. 419)). Additionally, many companies implement hardware with proprietary or draft features.

Tableau 24.1 *Overview of Various WLAN Standards*

Name	Band (GHz)	Maximum Transmission Rate (Mbit/s)	Note
802.11 Legacy	2.4	2	Outdated; virtually no end

Name	Band (GHz)	Maximum Transmission Rate (Mbit/s)	Note
			devices available
802.11a	5	54	Less interference-prone
802.11b	2.4	11	Less common
802.11g	2.4	54	Widespread, backwards-compatible with 11b
802.11n	2.4 and/or 5	300	Common

802.11 Legacy cards are not supported by openSUSE®. Most cards using 802.11a, 802.11b, 802.11g and 802.11n are supported. New cards usually comply with the 802.11n standard, but cards using 802.11g are still available.

24.2 Operating Modes

In wireless networking, various techniques and configurations are used to ensure fast, high-quality, and secure connections. Different operating types suit different setups. It can be difficult to choose the right authentication method. The available encryption methods have different advantages and pitfalls.

Basically, wireless networks can be classified into three network modes:

Managed Mode (Infrastructure Mode), via Access Point

Managed networks have a managing element: the access point. In this mode (also referred to as infrastructure mode), all connections of the WLAN stations in the network run through the access point, which may also serve as a connection to an ethernet. To make sure only authorized stations can connect, various authentication mechanisms (WPA, etc) are used.

Ad-hoc Mode (Peer-to-Peer Network)

Ad-hoc networks do not have an access point. The stations communicate directly with each other, therefore an ad-hoc network is usually faster than a managed network. However, the transmission range and number of participating stations are greatly limited in ad-hoc networks. They also do not support WPA authentication. If you intend to use WPA security, you should not use Ad-Hoc_Mode.

Master Mode

In master mode your network card is used as the access point. It works only if your WLAN card supports this mode. Find out the details of your WLAN card on <http://linux-wless.passys.nl>.

24.3 Authentication

Because a wireless network is much easier to intercept and compromise than a wired network, the various standards include authentication and encryption methods. In the original version of the IEEE 802.11 standard, these are described under the term WEP (Wired Equivalent Privacy). However, because WEP has proven to be insecure (see Section 24.6.3, « Security » (p. 433)), the WLAN industry (joined under the name *Wi-Fi Alliance*) has defined an extension called WPA, which is supposed to eliminate the weaknesses of WEP. The later IEEE 802.11i standard includes WPA and some other authentication and encryption methods. IEEE 802.11i is also referred to as WPA2, because WPA is based on a draft version of 802.11i.

To make sure that only authorized stations can connect, various authentication mechanisms are used in managed networks:

None (Open)

An open system is a system that does not require authentication. Any station can join the network. Nevertheless, WEP encryption can be used, see Section 24.4, « Encryption » (p. 423).

Shared Key (according to IEEE 802.11)

In this procedure, the WEP key is used for the authentication. However, this procedure is not recommended, because it makes the WEP key more susceptible to attacks. All an attacker needs to do is to listen long enough to the communication between the station and the access point. During the authentication process, both sides exchange the same information, once in encrypted form and once in

unencrypted form. This makes it possible for the key to be reconstructed with suitable tools. Because this method makes use of the WEP key for the authentication and for the encryption, it does not enhance the security of the network. A station that has the correct WEP key can authenticate, encrypt, and decrypt. A station that does not have the key cannot decrypt received packets. Accordingly, it cannot communicate, regardless of whether it had to authenticate itself.

WPA-PSK (or WPA-Personal, according to IEEE 802.1x)

WPA-PSK (PSK stands for preshared key) works similarly to the Shared Key procedure. All participating stations as well as the access point need the same key. The key is 256 bits in length and is usually entered as a passphrase. This system does not need a complex key management like WPA-EAP and is more suitable for private use. Therefore, WPA-PSK is sometimes referred to as WPA « Home ».

WPA-EAP (or WPA-Enterprise, according to IEEE 802.1x)

Actually, WPA-EAP (Extensible Authentication Protocol) is not an authentication system but a protocol for transporting authentication information. WPA-EAP is used to protect wireless networks in enterprises. In private networks, it is scarcely used. For this reason, WPA-EAP is sometimes referred to as WPA « Enterprise ».

WPA-EAP needs a Radius server to authenticate users. EAP offers three different methods for connecting and authenticating to the server:

- Transport Layer Security (EAP-TLS): TLS authentication relies on the mutual exchange of certificates for both server and client. First, the server presents its certificate to the client where it is evaluated. If the certificate is considered valid, the client in turn presents its certificate to the server. While TLS is secure, it requires a working certification management infrastructure in your network. This infrastructure is rarely found in private networks.
- Tunneled Transport Layer Security (EAP-TTSL)
- Protected Extensible Authentication Protocol (EAP-PEAP): Both TTLS and PEAP are two-stage protocols. In the first stage, a secure connection is established and in the second the client authentication data is exchanged. They require far less certification management overhead than TLS, if any.

24.4 Encryption

There are various encryption methods to ensure that no unauthorized person can read the data packets that are exchanged in a wireless network or gain access to the network:

WEP (defined in IEEE 802.11)

This standard makes use of the RC4 encryption algorithm, originally with a key length of 40 bits, later also with 104 bits. Often, the length is declared as 64 bits or 128 bits, depending on whether the 24 bits of the initialization vector are included. However, this standard has some weaknesses. Attacks against the keys generated by this system may be successful. Nevertheless, it is better to use WEP than to not encrypt the network at all.

Some vendors have implemented the non-standard « Dynamic WEP ». It works exactly as WEP and shares the same weaknesses, except that the key is periodically changed by a key management service.

TKIP (defined in WPA/IEEE 802.11i)

This key management protocol defined in the WPA standard uses the same encryption algorithm as WEP, but eliminates its weakness. Because a new key is generated for every data packet, attacks against these keys are fruitless. TKIP is used together with WPA-PSK.

CCMP (defined in IEEE 802.11i)

CCMP describes the key management. Usually, it is used in connection with WPA-EAP, but it can also be used with WPA-PSK. The encryption takes place according to AES and is stronger than the RC4 encryption of the WEP standard.

24.5 Configuration with YaST

IMPORTANT : Security Risks in Wireless Networks

Unencrypted WLAN connections allow third parties to intercept all network data. Be sure to protect your network traffic by using one of the supported authentication and encryption methods.

Use the best possible encryption method your hardware allows. However, to use a certain encryption method, all devices in the network must support

this method, otherwise they cannot communicate with each other. For example, if your router supports both WEP and WPA but the driver for your WLAN card only supports WEP, WEP is the least common denominator you can use. But even a weak encryption with WEP is better than none at all. Refer to Section 24.4, « Encryption » (p. 423) and Section 24.6.3, « Security » (p. 433) for information.

To configure a wireless LAN with YaST, you need to define the following parameters:

IP Address

Use either a static IP address or let a DHCP server dynamically assign an IP address to the interface.

Operating Mode

Defines how to integrate your machine into a WLAN, depending on the network topology. For background information, refer to Section 24.2, « Operating Modes » (p. 420).

Network Name (ESSID)

Unique string identifying a network.

Authentication and Encryption Details

Depending on the authentication and encryption method your network uses, you need to enter one or more keys and/or certificates.

Several input options are available for entering the respective keys: *Passphrase*, *ASCII* (only available for WEP authentication methods), and *Hexadecimal*.

24.5.1 Deactivating NetworkManager

A WLAN card is usually detected during installation. If your machine is a mobile computer, NetworkManager is usually activated by default. If instead you want to configure your WLAN card with YaST, you need to deactivate NetworkManager first:

- 1 Start YaST as user `root`.
- 2 In the YaST Control Center, select *Network Devices > Network Settings* to open the *Network Settings* dialog.

If your network is currently controlled by NetworkManager, you see a warning message that the network settings cannot be edited by YaST.

- 3 To enable editing with YaST, leave the message with *OK* and on the *Global Options* tab, activate *Traditional Method with ifup*.
- 4 For further configuration, proceed with Section 24.5.2, « Configuration for Access Points » (p. 425) or Section 24.5.3, « Establishing an Ad-Hoc Network » (p. 429).

Otherwise confirm your changes with *OK* to write the network configuration.

24.5.2 Configuration for Access Points

In this section, learn how to configure your WLAN card to connect to an (external) access point or how to use your WLAN card as access point if your WLAN card supports this. For configuration of networks without an access point, refer to Section 24.5.3, « Establishing an Ad-Hoc Network » (p. 429).

Procédure 24.1 *Configuring Your WLAN Card for Using an Access Point*

- 1 Start YaST and open the *Network Settings* dialog.
- 2 Switch to the *Overview* tab where all network cards are listed that have been detected by the system. If you need more information about general network configuration, refer to Section 13.4, « Configuring a Network Connection with YaST » (p. 236).
- 3 Choose your wireless card from the list and click *Edit* to open the *Network Card Setup* dialog.
- 4 On the *Address* tab, configure whether to use a dynamic or a static IP address for the machine. Usually *Dynamic Address* with *DHCP* is fine.
- 5 Click *Next* to proceed to the *Wireless Network Card Configuration* dialog.
- 6 To use your WLAN card to connect to an access point, set the *Operating Mode* to *Managed*.

If however you want to use your WLAN card as access point, set the *Operating Mode* to *Master*. Note that not all WLAN cards support this mode.

REMARQUE : Using WPA-PSK or WPA-EAP

If you want to use WPA-PSK or WPA-EAP authentication modes, the operating mode must be set to *Managed*.

- 7** To connect to a certain network, enter the *Network Name (ESSID)*. Alternatively, click *Scan Network* and select a network from the list of available wireless networks.

All stations in a wireless network need the same ESSID for communicating with each other. If no ESSID is specified, your WLAN card automatically associates with the access point that has the best signal strength.

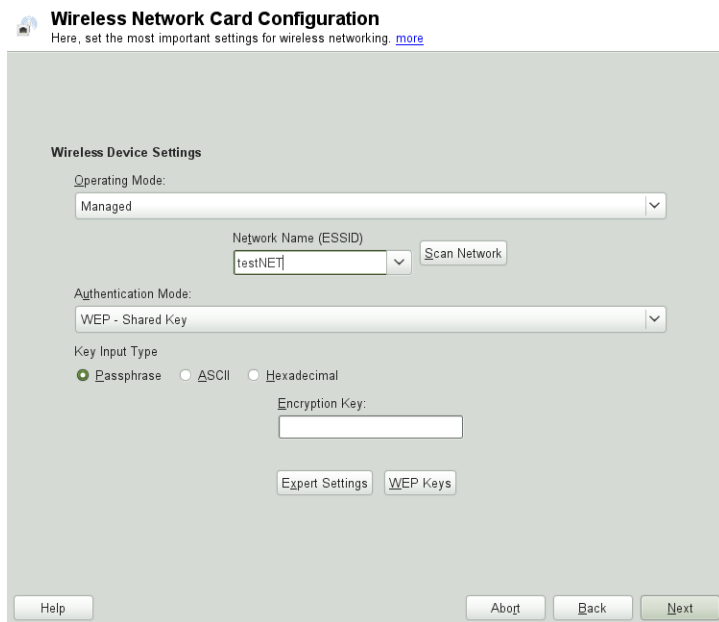
REMARQUE : WPA Authentication Requires an ESSID

If you select *WPA* authentication, a network name (ESSID) must be set.

- 8** Select an *Authentication Mode* for your network. Which mode is suitable, depends on your WLAN card's driver and the ability of the other devices in the network.
- 9** If you have chosen to set the *Authentication Mode* to *No Encryption*, finish the configuration by clicking *Next*. Confirm the message about this potential security risk and leave the *Overview* tab (showing the newly configured WLAN card) with *OK*.

If you haven chosen any of the other authentication modes, proceed with Procédure 24.2, « Entering the Encryption Details » (p. 427).

Figure 24.1 *YaST: Configuring the Wireless Network Card*



Procédure 24.2 *Entering the Encryption Details*

The following authentication methods require an encryption key: *WEP - Open*, *WEP - Shared Key*, and *WPA-PSK*.

For WEP, usually only key is needed—however, up to 4 different WEP keys can be defined for your station. One of them needs to be set as the default key and is used for encryption. The others are used for decryption. Per default, a key length of 128-bit is used, but you can also choose to set the length to 64-bit.

For higher security, WPA-EAP uses a RADIUS server to authenticate users. For authentication at the server, three different methods are available: TLS, TTLS and PEAP. The credentials and certificates you need for WPA-EAP depend on the authentication method used for the RADIUS server. Ask your system administrator to provide the needed information and credentials. YaST searches for any certificate under `/etc/cert`. Therefore, save the certificates given to you to this location and restrict access to these files to 0600 (owner read and write).

- 1 To enter the key for *WEP - Open* or *WEP - Shared Key*:

1a Set the *Key Input Type* either to *Passphrase*, *ASCII* or *Hexadecimal*.

1b Enter the respective *Encryption Key* (usually only one key is used):

If you have selected *Passphrase*, enter a word or a character string from which a key is generated according to the specified key length (per default, 128-bit) .

ASCII requests an input of 5 characters for a 64-bit key and 13 characters for a 128-bit key.

For *Hexadecimal*, enter 10 characters for a 64-bit key or 26 characters for a 128-bit key in hexadecimal notation.

1c To adjust the key length to a lower bit rate (which might be necessary for older hardware), click *WEP Keys* and set the *Key Length* to 64 bit. The *WEP Keys* dialog also shows the WEP keys that have been entered so far. Unless another key is explicitly set as default, YaST always uses the first key as default key.

1d To enter more keys for WEP or to modify one of the keys, select the respective entry and click *Edit*. Select the *Key Input Type* and enter the key.

1e Confirm your changes with *OK*.

2 To enter a key for *WPA-PSK*:

2a Select the input method *Passphrase* or *Hexadecimal*.

2b Enter the respective *Encryption Key*.

In the *Passphrase* mode, the input must be 8 to 63 characters. In the *Hexadecimal* mode, enter 64 characters.

3 If you have chosen *WPA-EAP* authentication, click *Next* to switch to the *WPA-EAP* dialog, where you enter the credentials and certificates you have been given by your network administrator.

3a Select the *EAP Mode* the RADIUS server uses for authentication. The details you need to enter in the following depend on the selected *EAP Mode*.

3b For TLS, provide *Identity*, *Client Certificate*, *Client Key*, and *Client Key Password*. To increase security, you can also configure a *Server Certificate* used to validate the server's authenticity.

TTLS and PEAP require *Identity* and *Password*, whereas *Server Certificate* and *Anonymous Identity* are optional.

3c To enter the advanced authentication dialog for your WPA-EAP setup, click *Details*.

3d Select the *Authentication Method* for the second stage of EAP-TTLS or EAP-PEAP communication (inner authentication). The choice of methods depends on the authentication method for the RADIUS server you selected in the previous dialog.

3e If the automatically-determined setting does not work for you, choose a specific *PEAP Version* to force the use of a certain PEAP implementation.

4 Confirm your changes with *OK*. The *Overview* tab shows the details of your newly configured WLAN card.

5 Click *OK* to finalize the configuration and to leave the dialog.

24.5.3 Establishing an Ad-Hoc Network

In some cases it is useful to connect two computers equipped with a WLAN card. To establish an ad-hoc network with YaST, do the following:

1 Start YaST and open the *Network Settings* dialog.

2 Switch to the *Overview* tab, choose your wireless card from the list and click *Edit* to open the *Network Card Setup* dialog.

3 Choose *Statically assigned IP Address* and enter the following data:

- *IP Address*: 192.168.1.1. Change this address on the second computer to 192.168.1.2, for example.
- *Subnet Mask*: /24

- *Hostname*: Choose any name you like.

4 Proceed with *Next*.

5 Set the *Operating Mode* to *Ad-hoc*.

6 Choose a *Network Name (ESSID)*. This can be any name, but it has to be used on every computer in the ad-hoc network.

7 Select an *Authentication Mode* for your network. Which mode is suitable, depends on your WLAN card's driver and the ability of the other devices in the network.

8 If you have chosen to set the *Authentication Mode* to *No Encryption*, finish the configuration by clicking *Next*. Confirm the message about this potential security risk and leave the *Overview* tab showing the newly configured WLAN card with *OK*.

If you haven't chosen any of the other authentication modes, proceed with Procedure 24.2, « Entering the Encryption Details » (p. 427).

9 If you do not have `smpppd` installed, YaST asks you to do so.

10 Configure the other WLAN cards in the network accordingly, using the same *Network Name (ESSID)*, the same *Authentication Mode* but different IP addresses.

24.5.4 Setting Additional Configuration Parameters

Usually there is no need to change the preconfigured settings when configuring your WLAN card. However, if you need detailed configuration of your WLAN connection, YaST allows you to tweak the following settings:

Channel

The specification of a channel on which the WLAN station should work. This is only needed in *Ad-hoc* and *Master* modes. In *Managed* mode, the card automatically searches the available channels for access points.

Bit Rate

Depending on the performance of your network, you may want to set a certain bit rate for the transmission from one point to another. In the default setting *Auto*, the system tries to use the highest possible data transmission rate. Some WLAN cards do not support the setting of bit rates.

Access Point

In an environment with several access points, one of them can be preselected by specifying the MAC address.

Power Management

When you are on the road, power saving technologies can help to maximize the operating time of your battery. Using power management may affect the connection quality and increase the network latency.

To access the advanced options:

- 1** Start YaST and open the *Network Settings* dialog.
- 2** Switch to the *Overview* tab, choose your wireless card from the list and click *Edit* to open the *Network Card Setup* dialog.
- 3** Click *Next* to proceed to the *Wireless Network Card Configuration* dialog.
- 4** Click *Expert Settings*.
- 5** In *Ad-hoc* mode, select one of the offered channels (11 to 14, depending on your country) for the communication of your station with the other stations. In *Master* mode, determine on which *Channel* your card should offer access point functionality. The default setting for this option is *Auto*.
- 6** Select the *Bit Rate* to use.
- 7** Enter the MAC address of the *Access Point* you want to connect to.
- 8** Choose if to *Use Power Management* or not.
- 9** Confirm your changes with *OK* and click *Next* and *OK* to finish the configuration.

24.6 Tips and Tricks for Setting Up a WLAN

The following tools and tips can help to monitor and improve speed and stability as well as security aspects of your WLAN.

24.6.1 Utilities

The package `wireless-tools` contains utilities that allow to set wireless LAN specific parameters and get statistics. See http://www.hpl.hp.com/personal/Jean_Tourrilhes/Linux/Tools.html for more information.

`kismet` (package `kismet`) is a network diagnosis tool with which to listen to the WLAN packet traffic. In this way, you can also detect any intrusion attempts in your network. More information is available at <http://www.kismetwireless.net/> and in the manual page.

24.6.2 Stability and Speed

The performance and reliability of a wireless network mainly depend on whether the participating stations receive a clear signal from the other stations. Obstructions like walls greatly weaken the signal. The more the signal strength sinks, the more the transmission slows down. During operation, check the signal strength with the `iwconfig` utility on the command line (Link Quality field) or with the NetworkManager applets provided by KDE or GNOME. If you have problems with the signal quality, try to set up the devices somewhere else or adjust the position of the antennas of your access points. Auxiliary antennas that substantially improve the reception are available for a number of PCMCIA WLAN cards. The rate specified by the manufacturer, such as 54 Mbit/s, is a nominal value that represents the theoretical maximum. In practice, the maximum data throughput is no more than half this value.

The `iwspy` command can displays WLAN statistics:

```
iwspy wlan0
wlan0      Statistics collected:
          00:AA:BB:CC:DD:EE : Quality:0   Signal level:0   Noise level:0
          Link/Cell/AP      : Quality:60/94 Signal level:-50 dBm   Noise
          level:-140 dBm (updated)
```

Typical/Reference : Quality:26/94 Signal level:-60 dBm Noise level:-90 dBm

24.6.3 Security

If you want to set up a wireless network, remember that anybody within the transmission range can easily access it if no security measures are implemented. Therefore, be sure to activate an encryption method. All WLAN cards and access points support WEP encryption. Although this is not entirely safe, it does present an obstacle for a potential attacker.

For private use, use WPA-PSK if available. Although Linux supports WPA on most hardware components, some drivers do not offer WPA support. It may also not be available on older access points and routers with WLAN functionality. For such devices, check if WPA can be implemented by means of a firmware update. If WPA is not available, WEP is better than no encryption. In enterprises with advanced security requirements, wireless networks should only be operated with WPA.

Use strong passwords for your authentication method. For example, the Web page <https://www.grc.com/passwords.htm> generates random 64 character passwords.

24.7 Troubleshooting

If your WLAN card is not automatically detected, check whether it is supported by openSUSE. A list of supported WLAN network cards is available under [http://en.opensuse.org/HCL:Network_\(Wireless\)](http://en.opensuse.org/HCL:Network_(Wireless)). If your card is not supported, it may be possible to make it work using the Microsoft Windows drivers with Ndiswrapper. Please refer to <http://en.opensuse.org/SDB:Ndiswrapper> for detailed information.

If your WLAN card fails to respond, check the following prerequisites:

1. Do you know the device name of the WLAN card? Usually it is `wlan0`. Check with the tool `ifconfig`.
2. Have you checked your needed firmware? Refer to `/usr/share/doc/packages/wireless-tools/README.firmware` for more information.
3. Is the ESSID of your router broadcasted and visible (not hidden)?

24.7.1 Check the Network Status

The command `iwconfig` can give you important information about your wireless connection. For example, the following line displays the ESSID, the wireless mode, frequency, if you signal is encrypted, the link quality, and much more:

```
iwconfig wlan0
wlan0 IEEE 802.11abg ESSID:"guest"
      Mode:Managed Frequency:5.22GHz Access Point: 00:11:22:33:44:55
      Bit Rate:54 Mb/s Tx-Power=13 dBm
      Retry min limit:7 RTS thr:off Fragment thr:off
      Encryption key:off
      Power Management:off
      Link Quality:62/92 Signal level:-48 dBm Noise level:-127 dBm
      Rx invalid nwid:0 Rx invalid crypt:0 Rx invalid frag:0
      Tx excessive retries:10 Invalid misc:0 Missed beacon:0
```

You can also get the previous information with the `iwlist` command. For example, the following line displays the current bit rate:

```
iwlist wlan0 rate
wlan0      unknown bit-rate information.
          Current Bit Rate=54 Mb/s
```

If you want an overview how many access points are available, it can also be done with the `iwlist` command. It gives you a list of « cells » which looks like this:

```
iwlist wlan0 scanning
wlan0 Scan completed:
      Cell 01 - Address: 00:11:22:33:44:55
              Channel:40
              Frequency:5.2 GHz (Channel 40)
              Quality=67/70 Signal level=-43 dBm
              Encryption key: off
              ESSID:"Guest"
              Bit Rates: 6 Mb/s; 9 Mb/s; 12 Mb/s; 18 Mb/s;
                        24 Mb/s; 36 Mb/s; 48 Mb/s
              Mode: Master
              Extra:tsf=0000111122223333
              Extra: Last beacon: 179ms ago
              IE: Unknown: ...
```

24.7.2 Multiple Network Devices

Modern laptops usually have a network card and a WLAN card. If you configured both devices with DHCP (automatic address assignment), you may encounter problems with the name resolution and the default gateway. This is evident from

the fact that you can ping the router but cannot surf the Internet. The Support Database features an article on this subject at http://old-en.opensuse.org/SDB:Name_Resolution_Does_Not_Work_with_Several_Concurrent_DHCP_Clients.

24.7.3 Problems with Prism2 Cards

Several drivers are available for devices with Prism2 chips. The various cards work more or less smoothly with the various drivers. With these cards, WPA is only possible with the hostap driver. If such a card does not work properly or not at all or you want to use WPA, read `/usr/share/doc/packages/wireless-tools/README.prism2`.

24.8 For More Information

More information can be found on the following pages:

http://www.hpl.hp.com/personal/Jean_Tourrilhes/Linux/Wireless.html

The Internet pages of Jean Tourrilhes, who developed the *Wireless Tools* for Linux, present a wealth of useful information about wireless networks.

tuxmobil.org

Useful hands-on information about mobile computers under Linux.

<http://www.linux-on-laptops.com>

More information about Linux on laptops.

[http://en.opensuse.org/HCL:Network_\(Wireless\)](http://en.opensuse.org/HCL:Network_(Wireless))

Lists supported WLAN network cards.

<http://en.opensuse.org/SDB:Ndiswrapper>

Offers a work-around for running unsupported WLAN cards with the Microsoft Windows using Ndiswrapper.

Using NetworkManager

NetworkManager is the ideal solution for laptops and other portable computers. It supports state-of-the-art encryption types and standards for network connections, including connections to 802.1X protected networks. 802.1X is the « IEEE Standard for Local and Metropolitan Area Networks—Port-Based Network Access Control ». With NetworkManager, you need not worry about configuring network interfaces and switching between wired or wireless networks when you are moving. NetworkManager can automatically connect to known wireless networks or manage several network connections in parallel—the fastest connection is then used as default. Furthermore, you can manually switch between available networks and manage your network connection using an icon in the system tray or in the top bar, respectively.

Instead of only one connection being active, multiple connections may be active at once. This enables you to unplug your laptop from an Ethernet and remain connected via a wireless connection.

25.1 Use Cases for NetworkManager

NetworkManager provides a sophisticated and intuitive user interface, which enables users to easily switch their network environment. However, NetworkManager is not a suitable solution in the following cases:

- Your computer provides network services for other computers in your network, for example, it is a DHCP or DNS server.

- Your computer is a Xen server or your system is a virtual system inside Xen.

25.2 Enabling or Disabling NetworkManager

On laptop computers, NetworkManager is enabled by default. However, it can be at any time enabled or disabled in the YaST Network Settings module.

- 1** Run YaST and go to *Network Devices > Network Settings*.
- 2** The *Network Settings* dialog opens. Go to the *Global Options* tab.
- 3** To configure and manage your network connections with NetworkManager:
 - 3a** In the *Network Setup Method* field, select *User Controlled with NetworkManager*.
 - 3b** Click *OK* and close YaST.
 - 3c** Configure your network connections with NetworkManager as described in Section 25.3, « Configuring Network Connections » (p. 439).
- 4** To deactivate NetworkManager and control network the traditional way:
 - 4a** In the *Network Setup Method* field, choose *Traditional Method with ifup*.
 - 4b** Click *OK*.
 - 4c** Set up your network card with YaST using automatic configuration via DHCP or a static IP address. Alternatively, configure your modem with YaST:
 - For dial-up connections, use *Network Devices > Modem*.
 - To configure an internal or USB ISDN modem, select *Network Devices > ISDN*.
 - To configure an internal or USB DSL modem, select *Network Devices > DSL*.

Find a detailed description of the network configuration with YaST in Section 13.4, « Configuring a Network Connection with YaST » (p. 236) and Chapitre 24, *Wireless LAN* (p. 419).

25.3 Configuring Network Connections

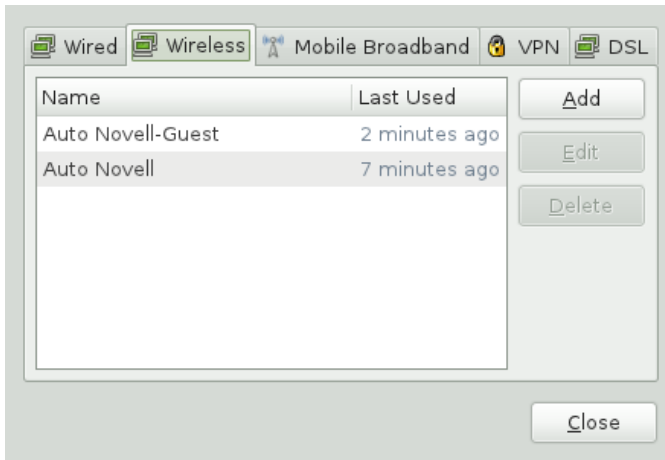
After having enabled NetworkManager in YaST, configure your network connections with the NetworkManager front-ends available in KDE and GNOME. The network configuration dialogs for both front-ends are very similar. They show tabs for all types of network connections, such as wired, wireless, mobile broadband, DSL, and VPN connections. On each tab, you can add, edit or delete connections of that type. If you hover the mouse pointer over the input fields and options, comprehensive tooltips are displayed. In the KDE configuration dialog, the appropriate tabs are only active if the connection type is available on your system (depending on hardware and software).

REMARQUE : Bluetooth Connections

Currently, Bluetooth connections cannot be configured with NetworkManager.

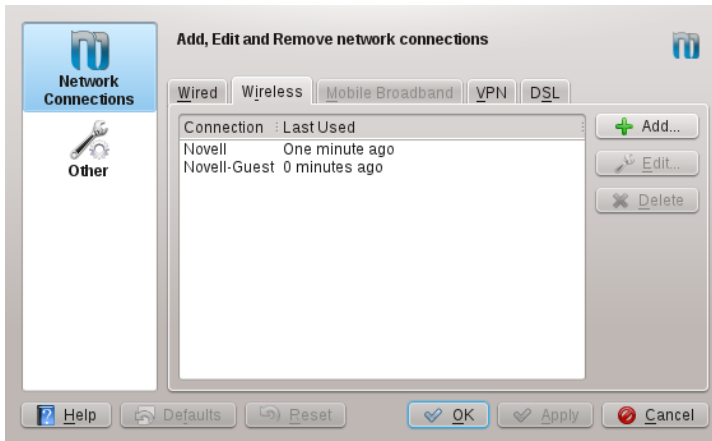
To open the network configuration dialog in GNOME, press Alt + F2 and enter `nm-connection-editor`.

Figure 25.1 *GNOME Network Connections Dialog*



If you are using KDE, open the main menu and click *System settings*. In the *Network and Connectivity* section select *Network Settings > Network Connections* to open the network configuration dialog.

Figure 25.2 *KDE Network Configuration Dialog*



Alternatively, you can also start the configuration dialogs from the NetworkManager front-end in the system tray or the top bar, respectively. In KDE, left-click the icon and select *Manage Connections*. In GNOME, click the icon and select *Network Settings > Options*.

REMARQUE : Availability of Options

Depending on your system setup, you may not be allowed to configure connections. In a secured environment, some options might be locked or require `root` permission. Ask your system administrator for details.

Procédure 25.1 *Adding or Editing Connections*

When configuring network connections with NetworkManager, you can also define `system connections` that can be shared by all users. In contrast to `user connections`, system connections are made available right after NetworkManager is started—before any users log in. For more details about both types of connections, refer to Section 25.7.1, « User and System Connections » (p. 450).

REMARQUE : Hidden Networks

To connect to a « hidden » network (a network that does not broadcast its service) you have to know the Service Set Identifier (SSID) or Extended Service Set Identifier (ESSID) of the network. Hidden networks cannot be detected automatically.

- 1** In the network configuration dialog, click the tab for the connection type you want to use.
- 2** Click *Add* to create a new connection or select an existing connection and click *Edit*.
- 3** Enter a *Connection Name* and your connection details.
- 4** You can tie the connection to a certain device, if more than one physical device per connection type is available (for example, your machine is equipped with two ethernet cards or two wireless cards).

If you are using KDE, do so by using the *Restrict to Interface* option. If you are using GNOME, enter the *MAC address* of the device you want to tie the connection to and confirm your settings.

- 5** For NetworkManager to automatically use a certain connection, activate the following option for this connection: *Connect Automatically* (KDE) or *Stay connected when possible* (GNOME).

- 6 To turn a connection into a system connection activate *System Connection* (KDE) or *Available to all users* (GNOME). To create and edit system connections, `root` permission is required.

After having confirmed your changes, the newly configured network connection appears in the list of available networks you get by left-clicking the NetworkManager icon.

Figure 25.3 KDE NetworkManager—Configured and Available Connections

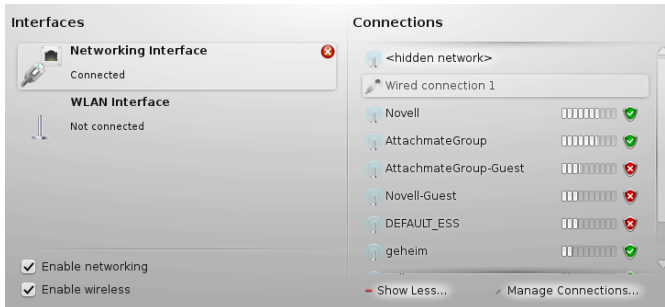
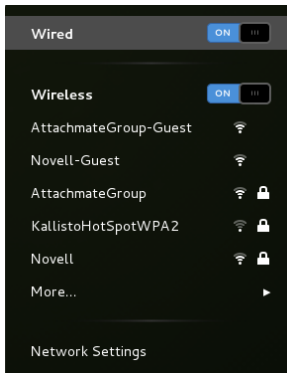


Figure 25.4 GNOME NetworkManager—Configured and Available Connections



25.4 Using the KDE NetworkManager Front-End

The KDE front-end for NetworkManager is the NetworkManager plasmoid. If the network has been set up for NetworkManager control, the plasmoid usually starts

automatically with the desktop environment and is shown as an icon in the system tray.

If your system tray does not show any network connection icon, the plasmoid is probably not started. Click the *Panel Tool Box* and choose *Add Widgets*. Double-click the *Network Management* entry and click the *Panel Tool Box* again.

The NetworkManager front-end only shows wireless networks for which you have configured a connection. It hides connections when you are out of range of a wireless network, or when the network cable is disconnected, thus always giving you a clear view of which connections may be used.

25.4.1 Managing Wired Network Connections

If your computer is connected to an existing network with a network cable, use the NetworkManager front-end to manage the network connection(s).

- 1** Left-click the NetworkManager icon to show a list with available *Interfaces* on the left-hand side and *Connections* on the right-hand side. The connections currently being used are shown in bold in the *Connections* list.
- 2** For detailed information and statistics for an interface, click the respective *Networking Interface* entry on the left-hand side of the plasmoid. Switch back to the interface overview by clicking the blue arrow icon.
- 3** To disconnect an active connection, click the red icon next to the respective *Networking Interface* entry.
- 4** If you want to use a different configuration with the wired network, click *Manage Connections* and add another wired connection as described in Procédure 25.1, « Adding or Editing Connections » (p. 441). Click the NetworkManager icon and select the newly configured connection to activate it.
- 5** To switch off all network connections, both wired and wireless, click the NetworkManager icon and uncheck *Enable Networking*.

25.4.2 Managing Wireless Network Connections

By default the NetworkManager front-end only lists connections that are already configured. The signal strength of each network is indicated by a series of bars, each one representing 10%. Encrypted wireless networks are marked with a green (WPA) or yellow (WEP) shield, while open networks are marked with a red shield.

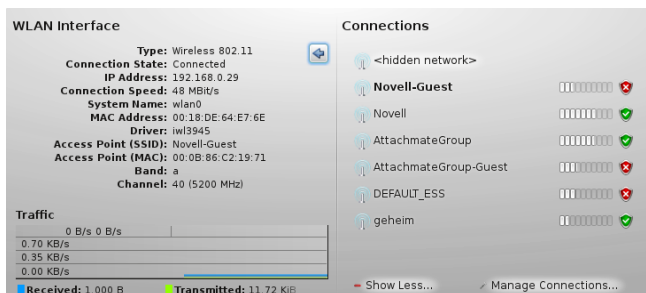
Procédure 25.2 *Connecting to a Wireless Network*

- 1 Click *Show More* to display all visible wireless networks available.
- 2 Left-click on the network of your choice to open the network connection dialog. To connect to a network that does not broadcast its service set identifier (SSID or ESSID), click *hidden network*. Fill in the connection details and press *Entrée*.
- 3 Based on your security settings for NetworkManager (see Section 25.7.2, « Storing Passwords and Credentials » (p. 451) for details), you may have to provide your KWallet password.
- 4 NetworkManager automatically connects to the configured network.

Procédure 25.3 *Managing Active Wireless Connections*

- 1 Click a *WLAN Interface* listed in the right half of the plasmoid window to get detailed information and statistics for that interface.

Figure 25.5 *KDE NetworkManager— Connection Details and Statistics*



Switch back to the interface overview by clicking on the blue arrow icon.

- 2 To disconnect an active connection, click the red icon for the *WLAN Interface*.
- 3 To completely disable wireless networking, uncheck *Enable Wireless*. This can be useful if you are on a plane or in any other environment where wireless networking is not allowed.

A wireless network that has been chosen explicitly will remain connected as long as possible. If a network cable is plugged in during that time, any connections that have been set to *Connect Automatically* will be connected, while the wireless connection remains up.

25.4.3 Configuring Your Wireless Card as an Access Point

If your wireless card supports access point mode, you can use NetworkManager for configuration.

REMARQUE : Availability of Options

Depending on your system setup, you may not be allowed to configure connections. In a secured environment, some options might be locked or require `root` permission. Ask your system administrator for details.

- 1 Start the dialog for configuring network connections as described in Section 25.3, « Configuring Network Connections » (p. 439).
- 2 Click *Add > Shared*.
- 3 On the *Wireless* tab provide a *Connection name* for the shared wireless network and a *SSID*.
- 4 Set the encryption on the *Wireless Security* tab.

IMPORTANT : Unprotected Wireless Networks Are a Security Risk

If you set *Security* to *None*, everybody can connect to your network, reuse your connectivity and intercept your network connection. To restrict access to your access point and to secure your connection, use encryption. You can choose between various WEP and WPA-based encryptions.

If you are not sure which technology is best for you, read Section 24.3, « Authentication » (p. 421).

5 Confirm your configuration with *OK*.

25.5 Using GNOME NetworkManager

In GNOME, NetworkManager can be controlled with the GNOME NetworkManager icon. If the network is set up for NetworkManager control, the icon is automatically shown in the top bar.

If your top bar does not show any network connection icon, GNOME NetworkManager is probably not started. Press **Alt + F2** and enter `nm-applet` to start it manually.

25.5.1 Managing Wired Network Connections

If your computer is connected to an existing network with a network cable, use the NetworkManager icon to choose the network connection.

- 1** Click the icon to show a menu with available networks. The currently used connection is listed on top of the menu, followed by other active connections.
- 2** Click *Network Settings* to get detailed information on all active interfaces.
- 3** To disconnect an active connection, click the *OFF* button next to *Wired*.
- 4** If you want to use a different configuration with the wired network, open the network configuration dialog and add another wired connection as described in Procédure 25.1, « Adding or Editing Connections » (p. 441). Click the NetworkManager icon and select the newly configured connection to activate it.
- 5** To switch off all network connections, both wired and wireless, right-click the icon and uncheck *Enable Networking*.

25.5.2 Managing Wireless Network Connections

GNOME NetworkManager lists a number of available visible wireless networks. To extend the list, click *More Networks*. The signal strength of each network is also shown in the menu. Encrypted wireless networks are marked with a lock icon.

Procédure 25.4 *Connecting to a Wireless Network*

- 1** To connect to a wireless network, click the NetworkManager icon and choose an entry from the list of available wireless networks.
- 2** If the network is encrypted, a dialog opens. It shows the type of encryption the network uses (*Wireless Security*) and holds a number of input fields according to the respective encryption and authentication settings. Enter the appropriate credentials.
- 3** To connect to a network that does not broadcast its service set identifier (SSID or ESSID), and therefore cannot be detected automatically, click the NetworkManager icon and choose *Network Settings > Wireless > Other*.
- 4** In the dialog that opens, enter the connection details and click *Connect*.
- 5** To disable wireless networking, click the icon and click the *OFF* button next to *Wireless*. This can be useful if you are on a plane or in any other environment where wireless networking is not allowed.

A wireless network that has been chosen explicitly will remain connected as long as possible. If a network cable is plugged in during that time, any connections that have been set to *Connect Automatically* will be connected, while the wireless connection remains up.

25.5.3 Configuring Your Wireless Card as an Access Point

If your wireless card supports access point mode, you can use NetworkManager for configuration.

REMARQUE : Availability of Options

Depending on your system set-up, you may not be allowed to configure connections. In a secured environment, some options might be locked or require `root` permission. Ask your system administrator for details.

- 1 Click the NetworkManager icon and select *Network Settings Wireless*.
- 2 Click *Use as Hotspot*, confirm the pop-up and enter the `root` password to continue.

A *Network Name* (SSID) and a *Security Key* are automatically generated and shown in the *Network* dialog. The network name will be based on the hostname of your computer. Other devices will need this information to connect to the hotspot that you have created.

IMPORTANT : Unprotected Wireless Networks Are a Security Risk

To restrict access to your access point and to secure your connection, use encryption. Depending on the abilities of your network card, you can choose between various WEP and WPA-based encryptions. If you are not sure which technology is best for you, read Section 24.3, « Authentication » (p. 421).

- 3 To change the SSID, encryption options (WEP, WPA, etc.) or the security key of the hotspot:
 - 3a Click *Options* next to the *Stop Hotspot* button.
 - 3b Enter the `root` password to continue.
 - 3c Change the SSID on the *Wireless* tab or the encryption details on the *Wireless Security* tab.

REMARQUE : WEP 40/128-bit Key (Hex or ASCII)

When using `WEP 40/128-bit Key` as encryption method, the *Key* length is restricted to either 5, 10, or 13 characters. Otherwise the *Save* button is inactive.

- 3d** Confirm your changes. After a short delay, the *Network* dialog will show your changes.
- 4** To stop the hotspot and disconnect any users, click *Stop Hotspot* and confirm your choice in the pop-up dialog.

25.6 NetworkManager and VPN

NetworkManager supports several Virtual Private Network (VPN) technologies. For each technology, openSUSE comes with a base package providing the generic support for NetworkManager. In addition to that, you also need to install the respective desktop-specific package for your front-end.

NovellVPN

To use this VPN technology, install

- `NetworkManager-novellvpn` and
- `NetworkManager-novellvpn-kde4` or `NetworkManager-novellvpn-gnome`.

NovellVPN support for KDE is not available yet, but is currently being worked on.

OpenVPN

To use this VPN technology, install

- `NetworkManager-openvpn` and
- `NetworkManager-openvpn-kde4` or `NetworkManager-openvpn-gnome`.

vpnc (Cisco)

To use this VPN technology, install

- `NetworkManager-vpnc` and
- `NetworkManager-vpnc-kde4` or `NetworkManager-vpnc-gnome`.

PPTP (Point-to-Point Tunneling Protocol)

To use this VPN technology, install

- `NetworkManager-pptp` and
- `NetworkManager-pptp-kde4` or `NetworkManager-pptp-gnome`.

After you have installed the packages, configure your VPN connection as described in Section 25.3, « Configuring Network Connections » (p. 439).

25.7 NetworkManager and Security

NetworkManager distinguishes two types of wireless connections, trusted and untrusted. A trusted connection is any network that you explicitly selected in the past. All others are untrusted. Trusted connections are identified by the name and MAC address of the access point. Using the MAC address ensures that you cannot use a different access point with the name of your trusted connection.

NetworkManager periodically scans for available wireless networks. If multiple trusted networks are found, the most recently used is automatically selected. NetworkManager waits for your selection in case that all networks are untrusted.

If the encryption setting changes but the name and MAC address remain the same, NetworkManager attempts to connect, but first you are asked to confirm the new encryption settings and provide any updates, such as a new key.

If you switch from using a wireless connection to offline mode, NetworkManager blanks the SSID or ESSID. This ensures that the card is disconnected.

25.7.1 User and System Connections

NetworkManager knows two types of connections: `user` and `system` connections. User connections are connections that become available to NetworkManager when the first user logs in. Any required credentials are asked from the user and when the user logs out, the connections are disconnected and removed from NetworkManager. Connections that are defined as system connection can be shared by all users and are made available right after NetworkManager is started—before any users log in. In case of system connections, all credentials must be provided at the time the connec-

tion is created. Such system connections can be used to automatically connect to networks that require authorization. For information how to configure user or system connections with NetworkManager, refer to Section 25.3, « Configuring Network Connections » (p. 439).

25.7.2 Storing Passwords and Credentials

If you do not want to re-enter your credentials each time you want to connect to an encrypted network, you can use the desktop-specific tools GNOME Keyring Manager or KWalletManager to store your credentials encrypted on the disk, secured by a master password.

In KDE, you can configure if and how to store your credentials. To do so, left-click the NetworkManager icon and select *Manage Connections*. Click *Other > Connection Secrets* and select one of the following options:

Do Not Store (Always Prompt)

This is useful if you are working in an environment where storing credentials is considered a security risk.

In File (Unencrypted)

If you choose this option, your passwords are stored unencrypted in the respective connection file that is created for each connection.

AVERTISSEMENT : Security Risk

Storing your network credentials unencrypted is a security risk. Everybody who has access to your computer can reuse your connectivity and intercept your network connection.

In Secure Storage (Encrypted)

If you choose this option, your credentials are stored in KWalletManager.

25.8 Frequently Asked Questions

In the following, find some frequently asked questions about configuring special network options with NetworkManager.

How to tie a connection to a specific device?

By default, connections in NetworkManager are device type-specific: they apply to all physical devices with the same type. If more than one physical device per connection type is available (for example, your machine is equipped with two ethernet cards), you can tie a connection to a certain device.

To do so in GNOME, first look up the MAC address of your device by using the output of the command line tool `ifconfig`. Then start the dialog for configuring network connections and choose the connection you want to modify. On the *Wired* or *Wireless* tab, enter the *MAC Address* of the device and confirm your changes.

If you are using KDE, start the dialog for configuring network connections and choose the connection you want to modify. On the *Ethernet* or *Wireless* tab, use the *Restrict to Interface* option to select the network interface to which to tie the connection.

How to specify a certain access point in case multiple access points with the same ESSID are detected?

When multiple access points with different wireless bands (a/b/g/n) are available, the access point with the strongest signal is automatically chosen by default. To override this, use the *BSSID* field when configuring wireless connections.

The Basic Service Set Identifier (BSSID) uniquely identifies each Basic Service Set. In an infrastructure Basic Service Set, the BSSID is the MAC address of the wireless access point. In an independent (ad-hoc) Basic Service Set, the BSSID is a locally administered MAC address generated from a 46-bit random number.

Start the dialog for configuring network connections as described in Section 25.3, « Configuring Network Connections » (p. 439). Choose the wireless connection you want to modify and click *Edit*. On the *Wireless* tab, enter the BSSID.

How to share network connections to other computers?

The primary device (the device which is connected to the Internet) does not need any special configuration. However, you need to configure the device that is connected to the local hub or machine as follows:

1. Start the dialog for configuring network connections as described in Section 25.3, « Configuring Network Connections » (p. 439). Choose the connection you want to modify and click *Edit*. If you are using GNOME, switch to the *IPv4 Settings* tab and from the *Method* drop-down list, choose

Shared to other computers. If you are using KDE, switch to the *IPv4 Address* or *IPv4 Address* tab and from the *Method* drop-down list, choose *Shared*. That will enable IP traffic forwarding and run a DHCP server on the device. Confirm your changes in NetworkManager.

2. As the DHCP server uses port 67, make sure that it is not blocked by the firewall: On the machine sharing the connections, start YaST and select *Security and Users > Firewall*. Switch to the *Allowed Services* category. If *DCHPv4 Server* is not already shown as *Allowed Service*, select *DCHPv4 Server* from *Services to Allow* and click *Add*. Confirm your changes in YaST.

How to provide static DNS information with automatic (DHCP, PPP, VPN) addresses?

In case a DHCP server provides invalid DNS information (and/or routes), you can override it. Start the dialog for configuring network connections as described in Section 25.3, « Configuring Network Connections » (p. 439). Choose the connection you want to modify and click *Edit*. If you are using GNOME, switch to the *IPv4 Settings* tab, and from the *Method* drop-down list, choose *Automatic (DHCP) addresses only*. If you are using KDE, switch to the *IPv4 Address* or *IPv4 Address* tab, and from the *Method* drop-down list, choose *Automatic (DHCP) addresses only*. Enter the DNS information in the *DNS Servers* and *Search Domains* fields. From the drop-down list at the top of the tab, select *Routes* if you want to *Ignore automatically obtained routes*. Confirm your changes.

How to make NetworkManager connect to password protected networks before a user logs in?

Define a `system connection` that can be used for such purposes. For more information, refer to Section 25.7, « NetworkManager and Security » (p. 450).

25.9 Troubleshooting

Connection problems can occur. Some common problems related to NetworkManager include the front-end not starting or a missing VPN option. Methods for resolving and preventing these problems depend on the tool used.

NetworkManager Front-End Does Not Start

The GNOME and KDE NetworkManager front-ends start automatically if the network is set up for NetworkManager control. If the front-end does not start, check if NetworkManager is enabled in YaST as described in Section 25.2,

« Enabling or Disabling NetworkManager » (p. 438). Then make sure that the appropriate package for your desktop environment is also installed. If you are using KDE 4, the package is `plasmoid-networkmanagement`. For GNOME users the package is `NetworkManager-gnome`.

If the desktop front-end is installed but is not running for some reason, start it manually with `Alt + F2` and `nm-applet` (GNOME) or by clicking the *Panel Tool Box* and choosing *Add Widgets*. (KDE) Double-click the *Network Management* entry and click the *Panel Tool Box* again.

NetworkManager Front-End Does Not Include the VPN Option

Support for NetworkManager, front-ends, and VPN for NetworkManager is distributed in separate packages. If your NetworkManager front-end does not include the VPN option, check if the packages with NetworkManager support for your VPN technology are installed. For more information, see Section 25.6, « NetworkManager and VPN » (p. 449).

No Network Connection Available

If you have configured your network connection correctly and all other components for the network connection (router, etc.) are also up and running, it sometimes helps to restart the network interfaces on your computer. To do so, log in to a command line as `root` and run `rcnetwork restart`.

25.10 For More Information

More information about NetworkManager can be found on the following Web sites and directories:

NetworkManager Project Page

<http://projects.gnome.org/NetworkManager/>

KDE NetworkManager Front-End

<http://userbase.kde.org/NetworkManagement>

Package Documentation

Also check out the information in the following directories for the latest information about NetworkManager and the GNOME and KDE NetworkManager front-ends:

- `/usr/share/doc/packages/NetworkManager/`,

- `/usr/share/doc/packages/NetworkManager-gnome/`.

Using Tablet PCs

openSUSE® comes with support for Tablet PCs. In the following, learn how to install and configure your Tablet PC and discover some useful Linux* applications which accept input from digital pens.

The following Tablet PCs are supported:

- Tablet PCs with serial and USB Wacom tablet (pen based), touch-screen or multi-touch devices.
- Tablet PCs with FinePoint devices, such as Gateway C210X/M280E/CX2724 or HP Compaq TC1000.
- Tablet PCs with touch screen devices, such as Asus R2H, Clevo TN120R, Fujitsu Siemens Computers P-Series, LG C1, Samsung Q1/Q1-Ultra.

After you have installed the Tablet PC packages and configured your digitizer correctly, input with the pen (also called a stylus) can be used for the following actions and applications:

- Logging in to KDM or GDM
- Unlocking your screen on the KDE and GNOME desktops
- Actions that can also be triggered by other pointing devices (such as mouse or touch pad), for example, moving the cursor on the screen, starting applications, closing, resizing and moving windows, shifting window focus and dragging and dropping objects

- Using gesture recognition in applications of the X Window System
- Drawing with GIMP
- Taking notes or sketching with applications like Jarnal or Xournal or editing larger amounts of text with Dasher

26.1 Installing Tablet PC Packages

The packages needed for Tablet PCs are included in the `TabletPC` installation pattern—if this is selected during installation, the following packages should already be installed on your system:

- `cellwriter`: a character-based handwriting input panel
- `jarnal`: a Java-based note taking application
- `xournal`: an application for note taking and sketching
- `xstroke`: a gesture recognition program for the X Window System
- `xvkbd`: a virtual keyboard for the X Window System
- `x11-input-fujitsu`: the X input module for Fujitsu P-Series tablets
- `x11-input-evtouch`: the X input module for some Tablet PCs with touch screens
- `xorg-x11-driver-input`: the X input module for input devices, including the module for Wacom devices.

If these packages are not installed, manually install the packages you need from command line or select the `TabletPC` pattern for installation in YaST.

26.2 Configuring Your Tablet Device

During installation, your tablet or touch device is configured by default. If you have trouble with the configuration of your Wacom device, you use `xsetwacom` on the command line to change the settings.

26.3 Using the Virtual Keyboard

To log in to the KDE or GNOME desktop or to unlock the screen, you can either enter your username and password as usual or via the virtual keyboard (xvkbd) displayed below the login field. To configure the keyboard or to access the integrated help, click the *xvkbd* field at the left lower corner and open the xvkbd main menu.

If your input is not visible (or is not transferred to the window where you need it), redirect the focus by clicking the *Focus* key in xvkbd and then clicking into the window that should get the keyboard events.

Figure 26.1 *xvkbd Virtual Keyboard*



If you want to use xvkbd after login, start it from the main menu or with *xvkbd* from a shell.

26.4 Rotating Your Display

Use *KRandRTray* (KDE) or *gnome-display-properties* (GNOME) to rotate or resize your display manually on the fly. Both *KRandRTray* and *gnome-display-properties* are applets for the *RANDR* extension of the X server.

Start *KRandRTray* or *gnome-display-properties* from the main menu, or enter *kran-drtray* or *gnome-display-properties* to start the applet from a shell. After you have started the applet, the applet icon is usually added to your system tray. If the *gnome-display-properties* icon does not automatically appear in the system tray, make sure *Show Displays in Panel* is activated in the *Monitor Resolution Settings* dialog.

To rotate your display with *KRandRTray*, right-click the icon and select *Configure Display*. Select the desired orientation from the configuration dialog.

To rotate your display with `gnome-display-properties`, right-click the icon and select the desired orientation. Your display is immediately tilted to the new direction. The orientation of the graphics tablet changes also, so it can still interpret the movement of the pen correctly.

If you have problems changing the orientation of your desktop, refer to Section 26.7, « Troubleshooting » (p. 464) for more information.

26.5 Using Gesture Recognition

openSUSE includes both `CellWriter` and `xstroke` for gesture recognition. Both applications accept gestures executed with the pen or other pointing devices as input for applications on the X Window System.

26.5.1 Using CellWriter

With `CellWriter`, you can write characters into a grid of cells—the writing is instantly recognized on a character basis. After you have finished writing, you can send the input to the currently focused application. Before you can use `CellWriter` for gesture recognition, the application needs to be trained to recognize your handwriting: You need to train each character of a certain map of keys (untrained characters are not activated and thus cannot be used).

Procédure 26.1 *Training CellWriter*

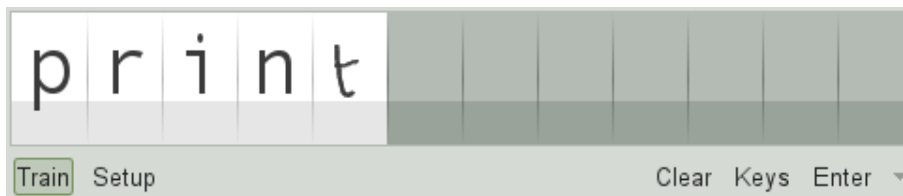
- 1 Start `CellWriter` from the main menu or with `cellwriter` from the command line. On the first start, `CellWriter` automatically starts in the training mode. In training mode it shows a set of characters of the currently chosen key map.
- 2 Enter the gesture you would like to use for a character into the respective character's cell. With the first input, the background changes its color to white, whereas the character itself is shown in light gray. Repeat the gesture multiple times until the character changes its color to black. Untrained characters are shown on a light gray or brown background (depending on the desktop's color scheme).
- 3 Repeat this step until you have trained `CellWriter` for all characters you need.
- 4 If you want to train `CellWriter` for another language, click the *Setup* button and select a language from the *Languages* tab. *Close* the configuration dialog. Click the

Train button and select the key map from the drop-down box at the bottom right corner of the *CellWriter* window. Now repeat your training for the new map of keys.

- 5 After having finished the training for the map of keys, click the *Train* button to switch to the normal mode.

In the normal mode, the *CellWriter* window shows a couple of empty cells in which to enter the gestures. The characters are not sent to another application until you click the *Enter* button, so you can correct or delete characters before you use them as input. Characters that have been recognized with a low degree of confidence will appear highlighted. To correct your input, use the context menu that appears on right-clicking a cell. To delete a character, either use your pen's eraser, or middle-click with the mouse to clear the cell. After finishing your input in *CellWriter*, define which application should receive the input by clicking into the application's window. Then send the input to the application by clicking *Enter*.

Figure 26.2 *Gesture Recognition with CellWriter*



If you click the *Keys* button in *CellWriter*, you get a virtual keyboard that can be used instead of the handwriting recognition.

To hide *CellWriter*, close the *CellWriter* window. The application now appears as icon in your system tray. To show the input window again, click the icon in the system tray.

26.5.2 Using Xstroke

With *xstroke*, you can use gestures with your pen or other pointing devices as input for applications on the X Window System. The *xstroke* alphabet is a unistroke alphabet that resembles the Graffiti* alphabet. When activated, *xstroke* sends the input to the currently focused window.

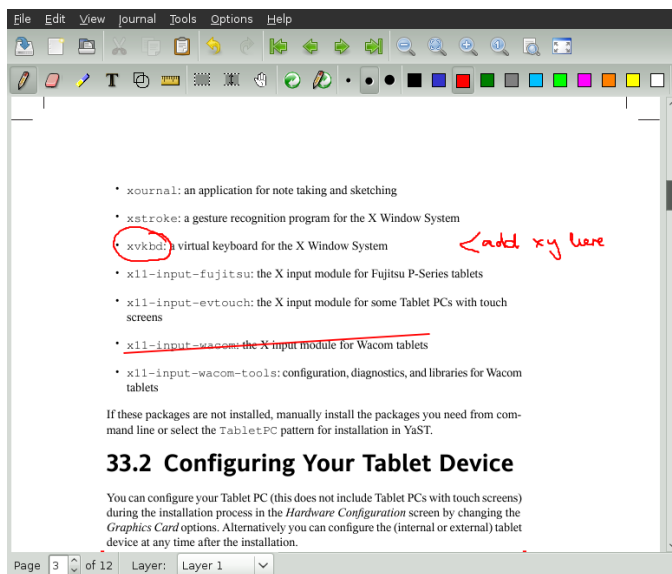
- 1 Start `xstroke` from the main menu or with `xstroke` from a shell. This adds a pencil icon to your system tray.
- 2 Start the application for which you want to create text input with the pen (for example, a terminal window, a text editor or an LibreOffice Writer).
- 3 To activate the gesture recognition mode, click the pencil icon once.
- 4 Perform some gestures on the graphics tablet with the pen or another pointing device. `xstroke` captures the gestures and transfers them to text that appears in the application window that has the focus.
- 5 To switch focus to a different window, click the desired window with the pen and hold for a moment (or use the keyboard shortcut defined in your desktop's control center).
- 6 To deactivate the gesture recognition mode, click the pencil icon again.

26.6 Taking Notes and Sketching with the Pen

To create drawings with the pen, you can use a professional graphics editor like GIMP or try one of the note-taking applications, Xournal or Jarnal. With both Xournal and Jarnal, you can take notes, create drawings or comment PDF files with the pen. As a Java-based application available for several platforms, Jarnal also offers basic collaboration features. For more information, refer to <http://www.dklevine.com/general/software/tc1000/jarnal-net.htm>. When saving your contents, Jarnal stores the data in an archive format (*.jaj) that also contains a file in SVG format.

Start Jarnal or Xournal from the main menu or by entering `jarnal` or `xournal` in a shell. To comment a PDF file in Xournal, for example, select *File > Annotate PDF* and open the PDF file from your file system. Use the pen or another pointing device to annotate the PDF and save your changes with *File > Print to PDF*.

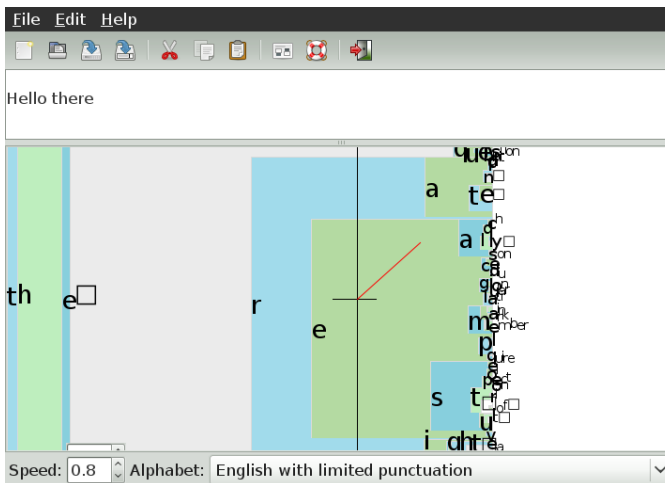
Figure 26.3 *Annotating a PDF with Xournal*



Dasher is another useful application. It was designed for situations where keyboard input is impractical or unavailable. With a bit of training, you can rapidly enter larger amounts of text using only the pen (or other input devices—it can even be driven with an eye tracker).

Start Dasher from the main menu or with `dasher` from a shell. Move your pen in one direction and the application starts to zoom into the letters on the right side. From the letters passing the cross hairs in the middle, the text is created or predicted and is printed to the upper part of the window. To stop or start writing, click the display once with the pen. Modify the zooming speed at the bottom of the window.

Figure 26.4 *Editing Texts with Dasher*



The Dasher concept works for many languages. For more information, refer to the Dasher Web site, which offers comprehensive documentation, demonstrations and training texts. Find it at <http://www.inference.phy.cam.ac.uk/dasher/>

26.7 Troubleshooting

Virtual Keyboard Does Not Appear on Login Screen

Occasionally, the virtual keyboard is not displayed on the login screen. To solve this, restart the X server by pressing `Ctrl + Alt + <—` or press the appropriate key on your Tablet PC (if you use a slate model without integrated keyboard). If the virtual keyboard still does not show, connect an external keyboard to your slate model and log in using the hardware keyboard.

Orientation of the Wacom Graphics Tablets Does Not Change

With the `xrandr` command, you can change the orientation of your display from within a shell. Enter `xrandr --help` to view the options available. To simultaneously change the orientation of your graphics tablet, the command needs to be modified as described below:

- For normal orientation (0° rotation):

```
xrandr -o normal && xsetwacom --set "Serial Wacom Tablet" Rotate  
NONE
```

- For 90° rotation (clockwise, portrait):

```
xrandr -o right && xsetwacom --set "Serial Wacom Tablet" Rotate  
CW
```

- For 180° rotation (landscape):

```
xrandr -o inverted && xsetwacom --set "Serial Wacom Tablet"  
Rotate HALF
```

- For 270° rotation (counterclockwise, portrait):

```
xrandr -o left && xsetwacom set --"Serial Wacom Tablet" Rotate  
CCW
```

Note that the commands above depend on the output of the `xsetwacom list` command. Replace "Serial Wacom Tablet" with the output for the stylus or the touch device. If you have a Wacom device with touch support (you can use your fingers on the tablet to move the cursor), you need to rotate also the touch device.

26.8 For More Information

Some of the applications mentioned here do not offer integrated online help, but you can find some useful information about usage and configuration in your installed system in `/usr/share/doc/package/packageName` or on the Web:

- For the Xournal manual, refer to <http://xournal.sourceforge.net/manual.html>
- The Jarnal documentation is located at <http://www.dklevine.com/general/software/tc1000/jarnal.htm#documentation>
- Find the xstroke man page at <http://davesource.com/Projects/xstroke/xstroke.txt>
- Find a HOWTO for configuring X on the Linux Wacom Web site: <http://linuxwacom.sourceforge.net/index.php/howto/x11>
- Find a very informative Web site about the Dasher project at <http://www.inference.phy.cam.ac.uk/dasher/>

- Find more information and documentation about CellWriter at <http://risujin.org/cellwriter/>
- Information on gnome-display-properties can be found at <http://old-en.opensuse.org/GNOME/Multiscreen>

Copying and Sharing Files

If using multiple operating systems (OS) simultaneously, it is often necessary to exchange files among them. Different systems may reside on different partitions on the same machine or on different machines across your network. There are various approaches to file exchange with different basic instructions and possible pitfalls.

AVERTISSEMENT : Scenarios for Private Home Networks Only

Do not use the following scenarios in networks other than your own private and trusted home network that is protected by a firewall. Implementing high security measures for the configurations featured in the following sections is beyond the scope of this document.

Exchanging data may encompass either one of the following tasks:

Copying

To copy your data means to transfer your data from one system to the other. This results in identical objects on both the source and the target system.

Synchronizing data is a special way to copy data. If you change a file on one computer, it is automatically changed on the other computer after the synchronization. For example, think of a laptop that contains your modified files and you want to have the same contents on your desktop computer.

Sharing

Sharing your files means establishing a client/server relationship. The server provides files that can be accessed by the client. When changing a file, you

modify it on the server, not locally on the client. File servers typically serve a large number of clients simultaneously.

27.1 Scenarios

The following list provides a number of possible scenarios involving file transfer:

Different OS on the Same Computer

Many users have an operating system preinstalled by their vendor and run Linux in a separate partition. Refer to Section 27.4, « Accessing Files on Different OS on the Same Computer » (p. 471) for more information.

Different Computers Not Connected by a Network

Save the data to any media (CD, DVD, USB flash drive, or external hard disk) and connect these to the target machine to copy your files. This solution is inexpensive, intuitive, and straightforward. However, you need the appropriate drives or ports on both computers. Additionally the operating systems have to understand the filesystem.

Media are suited to occasional file transfers with limited file size. If you need a more permanent solution, consider connecting them with a network.

Different Computers Connected to the Same Network

Set up a server of any kind on one computer, connect the server and the client, and transfer the files from server to client. Choose from various protocols available and pick the one that best matches your needs and expertise.

The client/server setup requires more expertise and maintenance efforts, but is better suited to routine transfer needs and exchange with multiple systems. If you are looking for a permanent file exchange, choose a client/server-based method. This method does not impose any limits on the amount of data that can be transferred. See Section 27.2, « Access Methods » (p. 469).

Different Computers on Different Networks

This scenario requires connection of different networks and is beyond the scope of this document. Transfer files as if the computers were not connected to a network.

27.2 Access Methods

The following methods and protocols are well-suited to file transfer and sharing.

FTP

Use FTP (File Transfer Protocol) if you need to exchange files very often and with different users. Set up an FTP server on one system and access it with clients. There are many graphical client applications available for FTP on Windows*, MacOS, and Linux. Depending on how your FTP server is used, enable read and write permissions. See Section 27.5.4, « Copying Files with FTP » (p. 478) for more details on FTP.

NFS

NFS (Network File System) is a client/server system. A server exports one or more directories that can be imported by a client. For more information, see Chapitre 18, *Sharing File Systems with NFS* (p. 325).

Use NFS if you share files very often and for different users. Generally, this protocol is more common in the Linux world than in the Windows world. An NFS export integrates well into your Linux system and you can browse the imported directory structure like any other folder on your local machine. Depending on your configuration, enable either read or write permissions or both on the server. In general, for a home user it makes sense to allow read and write access.

rsync

Use rsync to transfer regularly large volumes of data that does not change considerably. It is available on Linux and Windows. A typical use case for rsync is managing data backups. Refer to the manual page of the `rsync` command and Section 27.5.2, « Transferring Files with rsync » (p. 474) for more information.

Unison

Unison is an alternative to rsync. It is used to regularly synchronize files between different computers but has the advantage to behave bidirectionally. Refer to the manual page of the Unison command and Section 27.5.3, « Transferring Files with Unison » (p. 476) for more information. Unison is available on Linux and Windows.

CSync

CSync is an alternative to Unison. Just like Unison it synchronizes files bidirectionally. However, its architecture is modular so it can be extended with plug-ins. See <http://www.csync.org> for more details.

SMB

Samba is a client/server system and an implementation of the SMB protocol. It is usually used in Windows networks, but is supported by several operating systems. Refer to Chapitre 19, *Samba* (p. 339) for more information about Samba.

Use Samba if you need to share files very often and with different users, especially to Windows systems. Samba as a Linux-only solution is uncommon, use NFS instead. For more information about setting up a Samba server, refer to Section 27.8, « Sharing Files between Linux and Windows with Samba » (p. 484).

SSH

SSH (Secure Shell) enables a secure connection between computer. The SSH suite consists of several commands and uses public key encryption to authenticate users. For more information, see Chapitre 12, *SSH: Secure Network Operations* (↑Security Guide).

Use SSH if you copy files occasionally over an untrusted network and if you are the only user doing so. Although there are graphical user interfaces available, SSH is mainly considered a command line utility and is available on Linux and Windows.

27.3 Accessing Files Using a Direct Connection

This section describes one way to exchange files between two computers using an Ethernet crossover cable.

You need:

- Ethernet crossover cable. For further information see: http://en.wikipedia.org/wiki/Ethernet_crossover_cable
- openSUSE on both computers

- An established connection.
- The SSH daemon running on both machines. To start the service, run the command `rcsshd start as root`.

Proceed as follows:

Procédure 27.1 *GNOME*

- 1 Start Nautilus.
- 2 Click on *File > Connect to Server*.
- 3 Set the *Service Type* to *ssh*.
- 4 Enter the IP address and port of the remote computer (default is 22).
- 5 Specify the folder you want to open on the remote Computer.
- 6 Click *Connect*.

Procédure 27.2 *KDE*

- 1 Start Dolphin.
- 2 Click on *Network, Add Network*. Re-attach the pane if it is not available with *View > Panels > Places*.
- 3 Set the type of network to *Secure shell (ssh)*.
- 4 Enter any name and the correct user, IP address, port (default is 22) and folder of the remote Computer. It is also possible to create an icon for this connection by enabling the checkbox below. This connection icon appears in the *Network* tab in Dolphin.
- 5 Click on *Save & Connect* a dialog box opens and requests the password.

A new window containing the files of the remote computer will be opened.

27.4 Accessing Files on Different OS on the Same Computer

New computers generally ship with a preinstalled operating system, usually Windows. If you have installed Linux on a different partition, you might want to exchange files between the different operating systems.

Windows cannot read Linux partitions by default. If you want to exchange files between these two operating systems, you have to create an « exchange partition ». For a more direct approach, see <http://www.fs-driver.org/> to get a driver supporting an ext2 filesystem on Windows. The following file systems are used by Windows and can be accessed from a Linux machine:

FAT

Various flavors of this file system are used by MS-DOS and Windows 95 and 98. You can create this type of file system with YaST. It is possible to read and write files on FAT partitions from Linux. The size of a FAT partition (and even the maximum size of a single file) is subject to restrictions, depending on the FAT version. See <http://en.wikipedia.org/wiki/VFAT> for more information about FAT file systems.

NTFS

The NTFS file system is used by Windows. openSUSE includes write access support to the NTFS file system. See <http://en.opensuse.org/SDB:NTFS> for more information about NTFS-3g.

During the installation of openSUSE, your Windows partitions are detected. After starting your Linux system, the Windows partitions usually are mounted. These are possible ways of accessing your Windows data:

KDE

Press **Alt + F2** and enter `sysinfo:/`. A new window opens displaying the characteristics of your machine. *Disk Information* lists your partitions. Look at those that are of the file system type `ntfs` or `vfat` and click these entries. If the partition is not already mounted, KDE mounts the partition now and displays the contents.

Command Line

Just list the contents of `/windows` to see one or more directories containing your Windows drives. The directory `/windows/c` maps to the Windows `C:` \ drive, for example.

REMARQUE : Changing the Accessibility of Windows Partitions

Initially, Windows partitions are mounted read-only for normal users to avoid accidental damage to the file system. To grant normal users full access to a mounted Windows partition, change the mount behavior of this Windows partition. Refer to the manual page of the `mount` command for more information on mount options for `vfat` and to the manual page of `ntfs-3g` on mount options for NTFS.

27.5 Copying Files between Linux Computers

Linux offers a rich set of protocols you can use to copy files between computers. Which protocol you use depends on how much effort you want to invest and whether it needs to be compatible with future Windows installations. The following sections feature various methods to transfer files from and to Linux computers. Make sure that you have a working network connection, because otherwise they will not work. All scenarios rely on working name resolution in the network. If your network does not include a name service, use IP addresses directly or add the IP addresses along with respective hostnames to `/etc/hosts` on all clients.

The following example IP addresses and hostnames are used across this section:

Target Hostname	jupiter.example.com
Target IP	192.168.2.100
Source Hostname	venus.example.com
Source IP	192.168.2.101
User	tux

27.5.1 Copying Files with SSH

The following requirements must be met on both computers that are accessed via SSH:

1. If you use a hostname, make sure each hostname is listed in `/etc/hosts` on both computers (see Section 13.6.1.6, « `/etc/hosts` » (p. 265).) If you use SSH with IP addresses, you do not need to change anything.
2. If you use a firewall, open the SSH port. To do so, start YaST, and select *Security and Users > Firewall*. Go to *Allowed Services* and check whether *SSH* is displayed as part of the list. If this is not the case, select SSH from *Service to Allow* and click *Add*. Apply your changes and leave YaST with *Next* and *Finish*.

To copy files from one computer to another, you need to know where the files are located. For example, to copy the single file `/srv/foo_file` from computer `jupiter.example.com` to the current directory, use the following `scp` command (the dot represents the current directory as the copy target location):

```
scp tux@jupiter.example.com:/srv/foo_file .
```

To copy a whole directory structure, use the recursive mode of `scp`:

```
scp -r tux@jupiter.example.com:/srv/foo_directory .
```

If your network does not provide name resolution, use the server's IP address directly:

```
scp tux@192.168.2.100:/srv/foo_file .
```

If you do not know exactly where your files are, use the `sftp` command. Copying files in KDE or GNOME with SFTP is very simple. Proceed as follows:

- 1 Press **Alt + F2**.
- 2 Enter the following at the address prompt (correct it to your own values):

```
sftp://tux@jupiter.example.com
```
- 3 Confirm the question regarding of authenticity and enter the password of `tux` on `jupiter.example.com`.
- 4 Drag and drop the desired files or directories to your desktop or a local directory.

KDE provides another protocol called `fish` that can be used if `sftp` is not available. The use of this protocol is similar to `sftp`. Just replace the `sftp` protocol prefix of the URL with `fish`:

```
fish://tux@jupiter.example.com
```

27.5.2 Transferring Files with `rsync`

rsync is useful for archiving or copying data and can also be used as a daemon to provide directories to the network (see Procédure 27.3, « Advanced Setup for rsync Synchronization » (p. 475)).

Before using rsync to synchronize files and directories between different computers, make sure that the following requirements are met:

1. The package `rsync` is installed.
2. Identical users are available on both systems.
3. Enough disk space is available on the server.
4. If you want to benefit from rsync's full potential, make sure that `rsyncd` is installed on the system to use as the server.

27.5.2.1 rsync Basic Mode

The basic mode of operation of rsync does not require any special configuration. rsync mirrors complete directories onto another system. Its usage is not much different from a regular copying tool, such as `scp`. The following command creates a backup of the home directory of `tux` on a backup server called `jupiter`:

```
rsync -Hbaz -e ssh /home/tux/ tux@jupiter:backup
```

Use the following command to restore your backup (without option `-b`):

```
rsync -Haz -e ssh tux@jupiter:backup /home/tux/
```

27.5.2.2 rsync Daemon Mode

Start the `rsyncd` daemon on one of your systems to make use of the full functionality of rsync. In this mode, it is possible to create synchronization points (modules) that can be accessed without an account. To use the `rsyncd` daemon, proceed as follows:

Procédure 27.3 *Advanced Setup for rsync Synchronization*

- 1 Log in as `root` and install the `rsync` package.
- 2 Configure your synchronization points in `/etc/rsyncd.conf`. Add a point with its name in brackets and add the `path` keyword like in the following example:

```
[FTP]
path = /srv/ftp
comment = An Example
```

3 Start the `rsyncd` daemon as `root` with `rcrsyncd start`. To start the `rsync` service automatically during each system boot, run `insserv rsyncd`.

4 List all files located in the `/srv/ftp` directory (note the double colon):

```
rsync -avz jupiter::FTP
```

5 Initiate the transfer by providing a target directory (in this example, the current directory is represented by a dot):

```
rsync -avz jupiter::FTP .
```

By default, files are not deleted while synchronizing with `rsync`. To force file deletion, add the `--delete` option. To make sure that `--delete` does not accidentally remove newer files, use the `--update` option instead. Any conflicts that arise must be resolved manually.

27.5.3 Transferring Files with Unison

Before using Unison to synchronize files and directories between different computers, make sure that the following requirements are met:

1. The package `unison` is installed.
2. Enough disk space is available on your local and remote computer.
3. If you want to benefit from Unison's full potential, make sure that Unison is also installed and running on the remote computer.

In case you need help, run Unison with the `-doc topics` option to get a full list of available sections.

For permanent settings, Unison allows the creation of *profiles* that specify Unison preferences such as the directories (roots) to synchronize, which types of files to ignore, and other options. The profiles are stored as text files in `~/.unison` with the file extension `*.prf`.

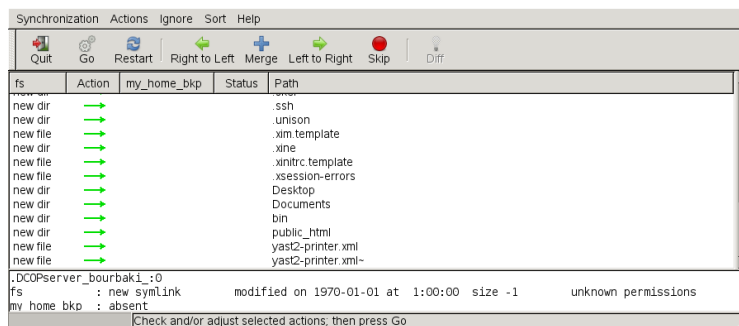
27.5.3.1 Using the GUI

To synchronize different directories with Unison's GUI, proceed as follows:

- 1 Start Unison by pressing **Alt + F2** and entering `unison`.
- 2 If you run Unison for the first time and without any further options, you are prompted for a source directory. Enter the source directory you want to synchronize and click *OK*.
- 3 Enter the target directory. It can be either local or remote. If you want to synchronize to a remote directory, choose the method (SSH, RSH or Socket) and enter the hostname and an optional user.
- 4 If you have not synchronized these two directories before, a warning dialog appears, informing you that Unison will now compare the contents of those directories. Close the warning with *OK* and wait until Unison has collected the information from both directories and displays the differences in the main window.

The left column shows the source directory you have selected, the third column shows the target directory. If there are differences between the directories, the *Action* column shows a symbol, proposing an action. A green arrow indicates that a file has been modified, added or deleted in the source or the target directory. The direction of the arrow indicates the direction that the change would be propagated if you performed the synchronization now. A question mark indicates a conflict (both files have been changed and Unison cannot decide which one to overwrite).

Figure 27.1 *File Synchronization Proposal*



- 5 To modify the proposals Unison shows for each file (for example, if you want to change the direction), select the file and click *Right to Left* or *Left to Right*. With *Skip*, exclude a file from synchronization. The symbol in the *Action* column changes accordingly.
- 6 To start the synchronization, click *Go*.

The next time you start Unison, a dialog box shows the existing profiles, each specifying a pair of directories to be synchronized. Select a profile or create a new profile (for another pair of directories) and perform the synchronization as described above.

27.5.3.2 Using the Command Line

Unison can also be operated through the command line. To synchronize a local directory to a remote computer, proceed as follows:

- 1 Open a shell and enter the following command:

```
unison -ui text DIR
          ssh://tux@jupiter.example.com//PATH
```

Replace the placeholders with the respective values.

- 2 Unison asks you what to do with your files and directories, for example:

```
local                jupiter
          <----  new file  dir [f]
```

- 3 Press F if you want to follow Unison's recommendation. For other commands, press ?.
- 4 Proceed with y, if you want to propagate your updates.

27.5.4 Copying Files with FTP

Before configuring your FTP server, make sure that the following requirements are met:

1. The package `vsftpd` is installed.
2. You have `root` access to your FTP server.
3. Enough disk space is available on your computer.

AVERTISSEMENT : For Home Networks Only

This setup is suited for use in home networks only. Do not deploy it to sites unprotected by firewalls and do not enable world wide access.

To configure an FTP server, proceed as follows:

1 Prepare the FTP server:

1a Open a shell, log in as `root`, and save a backup copy of `/etc/vsftpd.conf`:

```
cp /etc/vsftpd.conf /etc/vsftpd.conf.bak
```

1b Create an access point for anonymous FTP

```
mkdir ~ftp/incoming
chown -R ftp:ftp ~ftp/incoming
```

2 Replace the configuration files according to the preferred scenario (refer to the manual page of `vsftpd.conf` for advanced configuration options):

Allowing Anonymous Read and Write Access

```
#
listen=YES

# Enable anonymous access to FTP server
anonymous_enable=YES

#
local_enable=YES
# Enable write access
write_enable=YES
anon_upload_enable=YES
anon_mkdir_write_enable=YES
dirmessage_enable=YES
# Write log file
xferlog_enable=YES
connect_from_port_20=YES
chown_uploads=YES
chown_username=ftp
ftpd_banner=Welcome to FTP service.
anon_root=/srv/ftp
```

Grant Restricted Permissions to FTP Users (Home Only)

```
chroot_local_users=YES
```

3 Restart the FTP server:

```
rcvsftpd start
```

On the client, just enter the URL `ftp://HOST` in your browser or FTP client. Replace `HOST` with the hostname or IP address of your server. There are many

graphical user interfaces available that are suited to browsing the contents of your FTP server. For a list of them, just enter FTP at the search prompt of the YaST package manager.

27.6 Copying Files between Linux and Windows Computers with SSH

To transfer files from Linux to Windows using SSH, choose one of the following applications:

PuTTY

PuTTY is a suite of different command line tools for working with an SSH daemon. Download it from <http://www.chiark.greenend.org.uk/~sgtatham/putty.html>.

WinSCP

WinSCP is very similar to PuTTY, but includes a graphical user interface. Choose from an Explorer or Norton Commander style. Download it from <http://winscp.net>.

To copy a file from Windows to Linux with PuTTY, proceed as follows (on the Windows machine):

- 1 Start PSCP.
- 2 Enter the hostname of your SSH server.
- 3 Enter your login and password to the SSH server.

To connect from Windows to Linux with WinSCP, proceed as follows (on the Windows machine):

- 1 Start WinSCP.
- 2 Enter the hostname of the SSH server and username.
- 3 Click *Login* and acknowledge the resulting warning.

- 4 Drag and drop any files or directories from or to your WinSCP window.

REMARQUE : SSH Fingerprint

With both PuTTY and WinSCP, you must accept the SSH fingerprint when you log in for the first time.

27.7 Sharing Files between Linux Computers

The following sections feature various methods for sharing data. Use one of these if you are looking for a permanent solution for data sharing.

27.7.1 Transferring Files with NFS

To configure the server, proceed as follows:

- 1 Prepare the system:

- 1a Open a shell, log in as `root`, and grant write permissions to all users:

```
mkdir /srv/nfs
chgrp users /srv/nfs
chmod g+w /srv/nfs
```

- 1b Make sure that your user name and user ID is known on the client as well as on the server. Refer to Chapitre 10, *Managing Users with YaST* (↑Start-Up) for detailed instructions about how to create and manage user accounts.

- 2 Prepare the NFS server:

- 2a Start YaST as `root`.

- 2b Select *Network Services > NFS Server* (this module is not installed by default. If it is missing in YaST, install the package `yast2-nfs-server`).

- 2c Enable NFS services with *Start*.

2d Open the appropriate firewall port with *Open Port in Firewall* if you are using a firewall.

3 Export the directories:

3a Click *Add directory* and select `/srv/nfs`.

3b Set the export options to:

`rw,root_squash,async`

3c Repeat these steps, if you need to export more than one directory.

4 Apply your settings and leave YaST. Your NFS server is ready to use.

To manually start the NFS server, enter `rcnfsserver start` as `root`. To stop the server, enter `rcnfsserver stop`. By default, YaST takes care of starting this service at boot time.

To configure the client, proceed as follows:

1 Prepare the NFS client:

1a Start YaST as `root`.

1b Select *Network Services > NFS Client*.

1c Activate *Open Port in Firewall* if using a firewall.

2 Import the remote file system:

2a Click *Add*.

2b Enter the name or IP address of the NFS server or click *Choose* to automatically scan the network for NFS servers.

2c Enter the name of your remote file system or automatically choose it with *Select*.

2d Enter an appropriate mount point, for example `/mnt`. If you repeat this step with another exported file system, make sure you choose another mount point than `/mnt`.

2e Repeat these steps if you need to import more than one external directory.

3 Apply your settings and leave YaST. Your NFS client is ready to use.

To start the NFS client manually, enter `rcnfs start`.

REMARQUE : Consistent User Names

If your home network is used by just a small number of users, set up identical users manually on all machines. If, however, you need a larger consistent user base across a larger home network, consider using NIS or LDAP to manage user data. For further information, refer to Chapitre 3, *Using NIS* (↑Security Guide) and Chapitre 4, *LDAP—A Directory Service* (↑Security Guide).

27.7.2 Sharing Files with Samba

This section introduces various methods to access files on a Samba server. Both KDE and GNOME ship with graphical tools for working with Samba shares. There is also a command line tool for accessing Samba servers.

27.7.2.1 Accessing Shares with KDE and GNOME

Both the KDE and GNOME desktops can access Samba shares through their file browsers. To access your share, proceed as follows:

1 Press **Alt + F2** and enter `smb://jupiter.example.com/share`.

The syntax of this URL is `smb://HOST/SHARENAME` with *HOST* representing the hostname (`jupiter.example.com`) or IP address and *SHARENAME* representing the share. See Étape 3b (p. 485).

2 Log in with the username and password. The password is set in Étape 4 (p. 485) or just hit **Entrée** if no password is needed.

3 Drag and drop any files or directories from or to your window.

If you do not know your workgroup, enter `smb:/` to list all workgroups available in your network. The Smb4K tool (package `smb4k`) can also be used to display all workgroups in your network and mount them on demand.

27.7.2.2 Accessing Shares from the Command Line

If you prefer using the command line, use the `smbclient` command. To log in to your Samba server, run:

```
smbclient //jupiter/share -U tux
```

Omit the `-U` option if you are the current user `tux`. After logging in successfully, use some basic commands like `ls` (list contents), `mkdir` (create directory), `get` (download file), and `put` (upload file). Use `help` to display all commands. Refer to the manual page of `smbclient` for more information.

27.8 Sharing Files between Linux and Windows with Samba

Samba is the first choice for transferring files between Windows and Linux machines. These are the most common use cases for Samba:

Transferring Files from Linux to Windows with the SMB Scheme

In the easiest case you do not have to configure a Linux server. Use the `smb:` scheme. For more information, see Section 27.7.2.1, « Accessing Shares with KDE and GNOME » (p. 483). Make sure that your workgroup is identical on both systems and that your directories are shared.

Transferring Files from Windows to Linux Using a Server

Configure a Samba server on your Linux computer. See Procédure 27.4, « Setting Up a Samba Server » (p. 484).

ASTUCE : Using Default Registry Entries for Your Windows System

Some Windows versions (95, 98) require a small change in the registry for enabling a different password authentication method. Simplify this step by installing the `samba-doc` package and copy the file `/usr/share/doc/packages/samba/registry` to your Windows drive. Start Windows and incorporate the changes by double-clicking on this file.

Procédure 27.4 *Setting Up a Samba Server*

To set up a Samba server, do the following:

1 Prepare the Samba server:

1a Start YaST as `root`.

1b Install the `samba` package.

1c Create a directory (for example, `/srv/share`).

2 Create the server configuration:

2a Select *Network Services > Samba Server*.

2b Select one of the workgroups or enter a new one (for example, `Penguin`).

2c Check *Primary Domain Controller (PDC)*

2d Select *During Boot* if the Samba service should be started every time your computer boots. Otherwise select *Manually*.

2e Activate *Open Port in Firewall* if you use a firewall.

3 Create your Windows share:

3a Change to the *Shares* tab and click *Add*.

3b Enter a name and description. The *Share Name* is used for accessing the share from your clients. *Share Description* describes the purpose of the share.

3c Select your path (for example, `/srv/share`).

3d Proceed with *OK*.

3e Activate *Allow Users to Share Their Directories*.

4 Provide a password for all users that are allowed to use this service:

```
smbpasswd -a tux
```

For easier configuration, just hit **Entrée** to leave the password empty. Take into account that the usernames on your Windows and Linux computers are probably

different. Configuring a consistent user base for both Windows and Linux is beyond the scope of this document.

5 Start the Samba server:

```
rcnmb start
rcsmb start
```

To check if everything has been successfully configured, enter:

```
smbclient -L localhost
```

After you hit **Entrée**, you should get something like the following:

```
Anonymous login successful
Domain=[PENGUIN] OS=[Unix] Server=[Samba 3.0.22-11-SUSE-CODE10]

Sharename      Type      Comment
-----
share          Disk      Shared directory
netlogon       Disk      Network Logon Service
IPC$           IPC       IPC Service (Samba 3.0.22-11-SUSE-CODE10)
ADMIN$         IPC       IPC Service (Samba 3.0.22-11-SUSE-CODE10)
Anonymous login successful
Domain=[PENGUIN] OS=[Unix] Server=[Samba 3.0.22-11-SUSE-CODE10]

Server          Comment
-----
SUSE-DESKTOP    Samba 3.0.22-11-SUSE-CODE10

Workgroup       Master
-----
TUX-NET         jupiter
```

27.9 For More Information

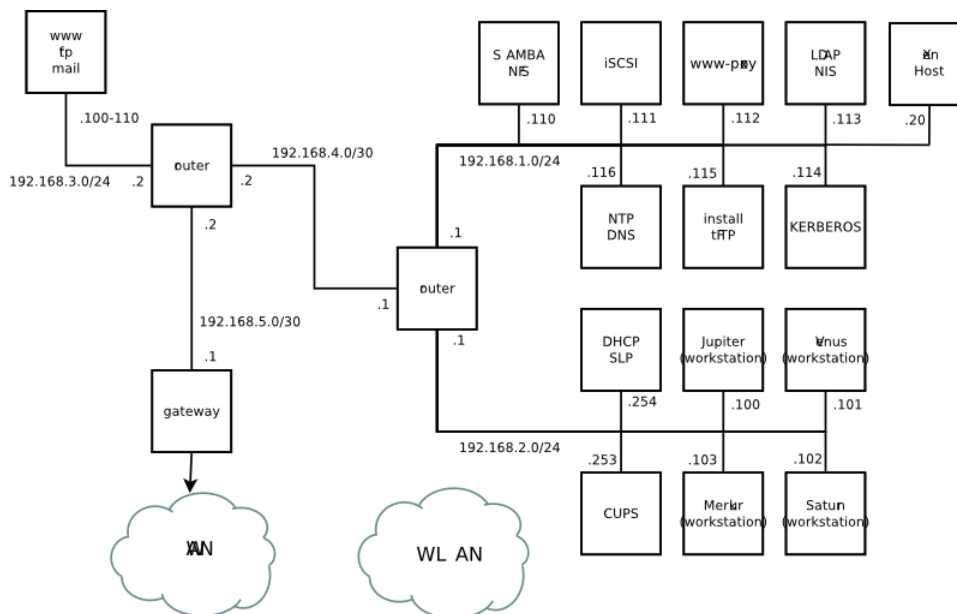
- <http://en.wikipedia.org/wiki/VFAT>
- <http://en.wikipedia.org/wiki/NTFS>
- <http://en.wikipedia.org/wiki/Fstab>
- http://en.wikipedia.org/wiki/Network_File_System
- http://en.wikipedia.org/wiki/File_Transfer_Protocol
- <http://en.wikipedia.org/wiki/SSH>

- <http://en.wikipedia.org/wiki/Rsync>
- http://en.wikipedia.org/wiki/Samba_software

A

An Example Network

This example network is used across all network-related chapters of the openSUSE® documentation.





Licences GNU

Cette annexe contient les licences GNU General Public License version 2 et GNU Free Documentation License version 1.2.

GNU General Public License

Version 2, June 1991

Copyright (C) 1989, 1991 Free Software Foundation, Inc. 59 Temple Place - Suite 330, Boston, MA 02111-1307, USA

Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

Preamble

The licenses for most software are designed to take away your freedom to share and change it. By contrast, the GNU General Public License is intended to guarantee your freedom to share and change free software--to make sure the software is free for all its users. This General Public License applies to most of the Free Software Foundation's software and to any other program whose authors commit to using it. (Some other Free Software Foundation software is covered by the GNU Library General Public License instead.) You can apply it to your programs, too.

When we speak of free software, we are referring to freedom, not price. Our General Public Licenses are designed to make sure that you have the freedom to distribute copies of free software (and charge for this service if you wish), that you receive source code or can get it if you want it, that you can change the software or use pieces of it in new free programs; and that you know you can do these things.

To protect your rights, we need to make restrictions that forbid anyone to deny you these rights or to ask you to surrender the rights. These restrictions translate to certain responsibilities for you if you distribute copies of the software, or if you modify it.

For example, if you distribute copies of such a program, whether gratis or for a fee, you must give the recipients all the rights that you have. You must make sure that they, too, receive or can get the source code. And you must show them these terms so they know their rights.

We protect your rights with two steps: (1) copyright the software, and (2) offer you this license which gives you legal permission to copy, distribute and/or modify the software.

Also, for each author's protection and ours, we want to make certain that everyone understands that there is no warranty for this free software. If the software is modified by someone else and passed on, we want its recipients to know that what they have is not the original, so that any problems introduced by others will not reflect on the original authors' reputations.

Finally, any free program is threatened constantly by software patents. We wish to avoid the danger that redistributors of a free program will individually obtain patent licenses, in effect making the program proprietary. To prevent this, we have made it clear that any patent must be licensed for everyone's free use or not licensed at all.

The precise terms and conditions for copying, distribution and modification follow.

GNU GENERAL PUBLIC LICENSE TERMS AND CONDITIONS FOR COPYING, DISTRIBUTION AND MODIFICATION

0. This License applies to any program or other work which contains a notice placed by the copyright holder saying it may be distributed under the terms of this General Public License. The « Program », below, refers to any such program or work, and a « work based on the Program » means either the Program or any derivative work under copyright law; that is to say, a work containing the Program or a portion of it, either verbatim or with modifications and/or translated into another language. (Hereinafter, translation is included without limitation in the term « modification ».) Each licensee is addressed as « you ».

Activities other than copying, distribution and modification are not covered by this License; they are outside its scope. The act of running the Program is not restricted, and the output from the Program is covered only if its contents constitute a work based on the Program (independent of having been made by running the Program). Whether that is true depends on what the Program does.

1. You may copy and distribute verbatim copies of the Program's source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice and disclaimer of warranty; keep intact all the notices that refer to this License and to the absence of any warranty; and give any other recipients of the Program a copy of this License along with the Program.

You may charge a fee for the physical act of transferring a copy, and you may at your option offer warranty protection in exchange for a fee.

2. You may modify your copy or copies of the Program or any portion of it, thus forming a work based on the Program, and copy and distribute such modifications or work under the terms of Section 1 above, provided that you also meet all of these conditions:

- a)** You must cause the modified files to carry prominent notices stating that you changed the files and the date of any change.
- b)** You must cause any work that you distribute or publish, that in whole or in part contains or is derived from the Program or any part thereof, to be licensed as a whole at no charge to all third parties under the terms of this License.
- c)** If the modified program normally reads commands interactively when run, you must cause it, when started running for such interactive use in the most ordinary way, to print or display an announcement including an appropriate copyright notice and a notice that there is no warranty (or else, saying that you provide a warranty) and that users may redistribute the program under these conditions, and telling the user how to view a copy of this License. (Exception: if the Program itself is interactive but does not normally print such an announcement, your work based on the Program is not required to print an announcement.)

These requirements apply to the modified work as a whole. If identifiable sections of that work are not derived from the Program, and can be reasonably considered independent and separate works in themselves, then this License, and its terms, do not apply to those sections when you distribute them as separate works. But when you distribute the same sections as part of a whole which is a work based on the Program, the distribution of the whole must be on the terms of this License, whose permissions for other licensees extend to the entire whole, and thus to each and every part regardless of who wrote it.

Thus, it is not the intent of this section to claim rights or contest your rights to work written entirely by you; rather, the intent is to exercise the right to control the distribution of derivative or collective works based on the Program.

In addition, mere aggregation of another work not based on the Program with the Program (or with a work based on the Program) on a volume of a storage or distribution medium does not bring the other work under the scope of this License.

3. You may copy and distribute the Program (or a work based on it, under Section 2) in object code or executable form under the terms of Sections 1 and 2 above provided that you also do one of the following:

- a)** Accompany it with the complete corresponding machine-readable source code, which must be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,
- b)** Accompany it with a written offer, valid for at least three years, to give any third party, for a charge no more than your cost of physically performing source distribution, a complete machine-readable copy of the corresponding source code, to be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,
- c)** Accompany it with the information you received as to the offer to distribute corresponding source code. (This alternative is allowed only for noncommercial distribution and only if you received the program in object code or executable form with such an offer, in accord with Subsection b above.)

The source code for a work means the preferred form of the work for making modifications to it. For an executable work, complete source code means all the source code for all modules it contains, plus any associated interface definition files, plus the scripts used to control compilation and installation of the executable. However, as a special exception, the source code distributed need not include anything that is normally distributed (in either source or binary form) with the major components (compiler, kernel, and so on) of the operating system on which the executable runs, unless that component itself accompanies the executable.

If distribution of executable or object code is made by offering access to copy from a designated place, then offering equivalent access to copy the source code from the same place counts as distribution of the source code, even though third parties are not compelled to copy the source along with the object code.

4. You may not copy, modify, sublicense, or distribute the Program except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense or distribute the Program 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.

5. You are not required to accept this License, since you have not signed it. However, nothing else grants you permission to modify or distribute the Program or its derivative works. These actions are prohibited by law if you do not accept this License. Therefore, by modifying or distributing the Program (or any work based on the Program), you indicate your acceptance of this License to do so, and all its terms and conditions for copying, distributing or modifying the Program or works based on it.

6. Each time you redistribute the Program (or any work based on the Program), the recipient automatically receives a license from the original licensor to copy, distribute or modify the Program subject to these terms and conditions. You may not impose any further restrictions on the recipients' exercise of the rights granted herein. You are not responsible for enforcing compliance by third parties to this License.

7. If, as a consequence of a court judgment or allegation of patent infringement or for any other reason (not limited to patent issues), conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot distribute so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not distribute the Program at all. For example, if a patent license would not permit royalty-free redistribution of the Program by all those who receive copies directly or indirectly through you, then the only way you could satisfy both it and this License would be to refrain entirely from distribution of the Program.

If any portion of this section is held invalid or unenforceable under any particular circumstance, the balance of the section is intended to apply and the section as a whole is intended to apply in other circumstances.

It is not the purpose of this section to induce you to infringe any patents or other property right claims or to contest validity of any such claims; this section has the sole purpose of protecting the integrity of the free software distribution system, which is implemented by public license practices. Many people have made generous contributions to the wide range of software distributed through that system in reliance on consistent application of that system; it is up to the author/donor to decide if he or she is willing to distribute software through any other system and a licensee cannot impose that choice.

This section is intended to make thoroughly clear what is believed to be a consequence of the rest of this License.

8. If the distribution and/or use of the Program is restricted in certain countries either by patents or by copyrighted interfaces, the original copyright holder who places the Program under this License may add an explicit geographical distribution limitation excluding those countries, so that distribution is permitted only in or among countries not thus excluded. In such case, this License incorporates the limitation as if written in the body of this License.

9. The Free Software Foundation may publish revised and/or new versions of the General Public 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.

Each version is given a distinguishing version number. If the Program specifies a version number of this License which applies to it and « any later version », you have the option of following the terms and conditions either of that version or of any later version published by the Free Software Foundation. If the Program does not specify a version number of this License, you may choose any version ever published by the Free Software Foundation.

10. If you wish to incorporate parts of the Program into other free programs whose distribution conditions are different, write to the author to ask for permission. For software which is copyrighted by the Free Software Foundation, write to the Free Software Foundation; we sometimes make exceptions for this. Our decision will be guided by the two goals of preserving the free status of all derivatives of our free software and of promoting the sharing and reuse of software generally.

NO WARRANTY

11. BECAUSE THE PROGRAM IS LICENSED FREE OF CHARGE, THERE IS NO WARRANTY FOR THE PROGRAM, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE PROGRAM "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE PROGRAM IS WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

12. IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MAY MODIFY AND/OR REDISTRIBUTE THE PROGRAM AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE PROGRAM (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE PROGRAM TO OPERATE WITH ANY OTHER PROGRAMS), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

END OF TERMS AND CONDITIONS

How to Apply These Terms to Your New Programs

If you develop a new program, and you want it to be of the greatest possible use to the public, the best way to achieve this is to make it free software which everyone can redistribute and change under these terms.

To do so, attach the following notices to the program. It is safest to attach them to the start of each source file to most effectively convey the exclusion of warranty; and each file should have at least the « copyright » line and a pointer to where the full notice is found.

one line to give the program's name and an idea of what it does.
Copyright (C) yyyy name of author

This program is free software; you can redistribute it and/or
modify it under the terms of the GNU General Public License
as published by the Free Software Foundation; either version 2
of the License, or (at your option) any later version.

This program is distributed in the hope that it will be useful,
but WITHOUT ANY WARRANTY; without even the implied warranty of
MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the
GNU General Public License for more details.

You should have received a copy of the GNU General Public License
along with this program; if not, write to the Free Software
Foundation, Inc., 59 Temple Place - Suite 330, Boston, MA 02111-1307, USA.

Also add information on how to contact you by electronic and paper mail.

If the program is interactive, make it output a short notice like this when it starts in an interactive mode:

```
Gnomovision version 69, Copyright (C) year name of author
Gnomovision comes with ABSOLUTELY NO WARRANTY; for details
type `show w'. This is free software, and you are welcome
to redistribute it under certain conditions; type `show c'
for details.
```

The hypothetical commands `show w' and `show c' should show the appropriate parts of the General Public License. Of course, the commands you use may be called something other than `show w' and `show c'; they could even be mouse-clicks or menu items--whatever suits your program.

You should also get your employer (if you work as a programmer) or your school, if any, to sign a « copyright disclaimer » for the program, if necessary. Here is a sample; alter the names:

```
Yoyodyne, Inc., hereby disclaims all copyright
interest in the program `Gnomovision'
(which makes passes at compilers) written
by James Hacker.
```

```
signature of Ty Coon, 1 April 1989
Ty Coon, President of Vice
```

This General Public License does not permit incorporating your program into proprietary programs. If your program is a subroutine library, you may consider it more useful to permit linking proprietary applications with the library. If this is what you want to do, use the GNU Lesser General Public License [<http://www.fsf.org/licenses/lgpl.html>] instead of this License.

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. Secondly, this License preserves for the author and publisher a way to get credit for their work, while not being considered responsible for modifications made by others.

This License is a kind of "copyleft", which means that derivative works of the document must themselves be free in the same sense. It complements the GNU General Public License, which is a copyleft license designed for free software.

We have designed this License 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 what-

soever 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.

