

Matthew Helmke

2021 Edition
Covers 20.04, 20.10, and 21.04

Ubuntu Linux

UNLEASHED



Matthew Helmke

with Andrew Hudson
and Paul Hudson

Ubuntu Linux

UNLEASHED

2021 Edition

SAMS |

Ubuntu Linux Unleashed 2021 Edition

Copyright © 2021 by Pearson Education, Inc.

All rights reserved. This publication is protected by copyright, and permission must be obtained from the publisher prior to any prohibited reproduction, storage in a retrieval system, or transmission in any form or by any means, electronic, mechanical, photocopying, recording, or likewise. For information regarding permissions, request forms, and the appropriate contacts within the Pearson Education Global Rights & Permissions Department, please visit www.pearson.com/permissions/. No patent liability is assumed with respect to the use of the information contained herein. Although every precaution has been taken in the preparation of this book, the publisher and author assume no responsibility for errors or omissions. Nor is any liability assumed for damages resulting from the use of the information contained herein.

ISBN-13: 978-0-13-677885-1

ISBN-10: 0-13-677885-2

Library of Congress Control Number: 2020937219

ScoutAutomatedPrintCode

Trademarks

All terms mentioned in this book that are known to be trademarks or service marks have been appropriately capitalized. Pearson Education cannot attest to the accuracy of this information. Use of a term in this book should not be regarded as affecting the validity of any trademark or service mark.

Microsoft and/or its respective suppliers make no representations about the suitability of the information contained in the documents and related graphics published as part of the services for any purpose. All such documents and related graphics are provided “as is” without warranty of any kind. Microsoft and/or its respective suppliers hereby disclaim all warranties and conditions with regard to this information, including all warranties and conditions of merchantability, whether express, implied or statutory, fitness for a particular purpose, title and non-infringement. In no event shall Microsoft and/or its respective suppliers be liable for any special, indirect or consequential damages or any damages whatsoever resulting from loss of use, data or profits, whether in an action of contract, negligence or other tortious action, arising out of or in connection with the use or performance of information available from the services. The documents and related graphics contained herein could include technical inaccuracies or typographical errors. Changes are periodically added to the information herein. Microsoft and/or its respective suppliers may make improvements and/or changes in the product(s) and/or the program(s) described herein at any time. Partial screen shots may be viewed in full within the software version specified.

Microsoft® and Windows® are registered trademarks of the Microsoft Corporation in the U.S.A. and other countries. Screenshots and icons reprinted with permission from the Microsoft Corporation. This book is not sponsored or endorsed by or affiliated with the Microsoft Corporation.

Warning and Disclaimer

Every effort has been made to make this book as complete and as accurate as possible, but no warranty or fitness is implied. The information provided is on an “as is” basis. The author and the publisher shall have neither liability nor responsibility to any person or entity with respect to any loss or damages arising from the information contained in this book.

Special Sales

For information about buying this title in bulk quantities, or for special sales opportunities (which may include electronic versions; custom cover designs; and content particular to your business, training goals, marketing focus, or branding interests), please contact our corporate sales department at corpsales@pearsoned.com or (800) 382-3419.

For government sales inquiries, please contact governmentsales@pearsoned.com.

For questions about sales outside the U.S., please contact intlcs@pearson.com.

Editor-in-Chief

Mark Taub

Acquisitions Editor

Debra Williams

Managing Editor

Sandra Schroeder

Senior Project Editor

Lori Lyons

Copy Editor

Paula Lowell

Production Manager

Aswini Kumar/
codeMantra

Indexer

Ken Johnson

Proofreader

Charlotte Kughen

Cover Designer

Chuti Prasertsith

Compositor

codeMantra

Contents at a Glance

Introduction	xxxi
Part I Getting Started	
1 Installing Ubuntu and Post-Installation Configuration	1
2 Background Information and Resources	23
Part II Desktop Ubuntu	
3 Foundations of the Linux GUI	31
4 Ubuntu Desktop Options	41
5 On the Internet	51
6 Productivity Applications	55
7 Multimedia Applications	63
8 Games	79
Part III System Administration	
9 Managing Software	91
10 Command-Line Beginner's Class	105
11 Command-Line Master Class, Part 1	141
12 Command-Line Master Class, Part 2	167
13 Managing Users	201
14 Automating Tasks and Shell Scripting	225
15 The Boot Process	271
16 System-Monitoring Tools	281
17 Backing Up	301
18 Networking	325
19 Remote Access with SSH and VNC	371
20 Securing Your Machines	379
21 Performance Tuning	393
22 Kernel and Module Management	401
Part IV Ubuntu as a Server	
23 Sharing Files and Printers	421
24 Common Web Server Stacks	437
25 Apache Web Server Management	443

26	Nginx Web Server Management.....	467
27	Other HTTP Servers.....	479
28	Administering Relational Database Services.....	483
29	NoSQL Databases.....	509
30	Virtualization on Ubuntu.....	521
31	Containers and Ubuntu.....	531
32	Ubuntu and Cloud Computing.....	535
33	Managing Sets of Servers.....	543
34	Handling Email.....	547
35	Proxying, Reverse Proxying, and Virtual Private Networks (VPNs).....	563
36	Lightweight Directory Access Protocol (LDAP).....	579
37	Name Serving with the Domain Name System (DNS).....	587
Part V Programming Linux		
38	Using Programming Tools.....	597
39	Using Popular Programming Languages.....	621
40	Helping with Ubuntu Development.....	635
41	Helping with Ubuntu Testing and QA.....	645
	Index.....	649

NOTE

Bonus Chapters 42–44 can be accessed online at informit.com/title/9780136778851.

Part VI Bonus Online Chapters

42	Using Perl.....	Web:1
43	Using Python.....	Web:21
44	Using PHP.....	Web:39

Table of Contents

Introduction

xxxi

Part I Getting Started

1	Installing Ubuntu and Post-Installation Configuration	1
	Before You Begin the Installation	1
	Researching Your Hardware Specifications	2
	Installation Options	2
	32-Bit Versus 64-Bit Ubuntu	4
	Planning Partition Strategies	5
	The Boot Loader	5
	Installing from DVD or USB Drive	6
	Step-by-Step Installation	6
	Installing	7
	First Update	11
	Shutting Down	11
	Finding Programs and Files	12
	Software Updater	12
	The <code>sudo</code> Command	14
	Configuring Software Repositories	15
	System Settings	17
	Detecting and Configuring a Printer	18
	Configuring Power Management in Ubuntu	18
	Setting the Time and Date	19
	Configuring Wireless Networks	20
	Troubleshooting Post-Installation Configuration Problems	21
	References	22
2	Background Information and Resources	23
	What Is Linux?	23
	Why Use Linux?	25
	What Is Ubuntu?	27
	Ubuntu for Business	27
	Ubuntu in Your Home	28
	Getting the Most from Linux and Ubuntu Documentation	28
	Linux	29
	Ubuntu	30

Part II Desktop Ubuntu

3	Foundations of the Linux GUI	31
	Foundations and the X Server	31
	Basic X Concepts	32
	Using X	33
	Elements of the <code>xorg.conf</code> File	34
	Starting X	39
	Using a Display Manager	39
	Changing Window Managers	39
	References	40
4	Ubuntu Desktop Options	41
	Desktop Environment	41
	Using GNOME: A Primer	42
	KDE and Kubuntu	45
	Xfce and Xubuntu	46
	LXDE and Lubuntu	47
	MATE and Ubuntu MATE	48
	Ubuntu Budgie	49
	Ubuntu Kylin	50
	References	50
5	On the Internet	51
	Getting Started with Firefox	52
	Checking Out Google Chrome and Chromium	53
	References	54
6	Productivity Applications	55
	Introducing LibreOffice	56
	Other Useful Productivity Software	58
	Working with PDFs	58
	Writing Scripts	59
	Working with XML and DocBook	59
	Working with LaTeX	60
	Creating Mind Maps	61
	Productivity Applications Written for Microsoft Windows	61
	References	61
7	Multimedia Applications	63
	Sound and Music	63
	Sound Cards	64
	Sound Formats	65
	Listening to Music	65

Graphics Manipulation	66
The GNU Image Manipulation Program	66
Using Scanners in Ubuntu	67
Working with Graphics Formats	67
Capturing Screen Images	69
Other Graphics Manipulation Options	70
Using Digital Cameras with Ubuntu	70
Handheld Digital Cameras	70
Using Shotwell Photo Manager	71
Burning CDs and DVDs in Ubuntu	71
Creating CDs and DVDs with Brasero	71
Creating CDs from the Command Line	72
Creating DVDs from the Command Line	73
Viewing Video	75
Video Formats	75
Viewing Video in Linux	76
Recording and Editing Audio	76
Editing Video	77
References	77
8 Games	79
Ubuntu Gaming	79
Installing Proprietary Video Drivers	80
Online Game Sources	81
Steam	81
GOG.com	82
Humble	82
itch.io	82
LGDB	82
Game Jolt	82
Installing Games from the Ubuntu Repositories	82
<i>Warsow</i>	82
<i>Scorched 3D</i>	83
<i>Frozen Bubble</i>	84
<i>SuperTux</i>	84
<i>Battle for Wesnoth</i>	85
<i>Frets on Fire</i>	85
<i>FlightGear</i>	87
<i>Speed Dreams</i>	87
Games for Kids	88
Commercial Games	88
Playing Windows Games	88
References	89

Part III System Administration

9	Managing Software	91
	Ubuntu Software	91
	Using Synaptic for Software Management	92
	Staying Up to Date	94
	Working on the Command Line	95
	Day-to-Day APT Usage	95
	Finding Software	98
	Using <code>apt-get</code> Instead of <code>apt</code>	99
	Compiling Software from Source	100
	Compiling from a Tarball	100
	Compiling from Source from the Ubuntu Repositories	101
	Configuration Management	102
	dotdeb	102
	Ubuntu Core	103
	Using the Snappy Package Manager	103
	References	104
10	Command-Line Beginner's Class	105
	What Is the Command Line?	106
	Accessing the Command Line	107
	Text-Based Console Login	107
	Logging Out	108
	Logging In and Out from a Remote Computer	108
	User Accounts	109
	Reading Documentation	111
	Using Man Pages	111
	Using <code>apropos</code>	112
	Using <code>whereis</code>	112
	Understanding the Linux File System Hierarchy	112
	Essential Commands in <code>/bin</code> and <code>/sbin</code>	114
	Configuration Files in <code>/etc</code>	114
	User Directories: <code>/home</code>	115
	Using the Contents of the <code>/proc</code> Directory to Interact with or Obtain Information from the Kernel	115
	Working with Shared Data in the <code>/usr</code> Directory	117
	Temporary File Storage in the <code>/tmp</code> Directory	117
	Accessing Variable Data Files in the <code>/var</code> Directory	117
	Navigating the Linux File System	117
	Listing the Contents of a Directory with <code>ls</code>	118
	Changing Directories with <code>cd</code>	120
	Finding Your Current Directory with <code>pwd</code>	120

Working with Permissions	120
Assigning Permissions	121
Directory Permissions	122
Altering File Permissions with <code>chmod</code>	123
File Permissions with <code>umask</code>	124
File Permissions with <code>chgrp</code>	125
Changing File Permissions with <code>chown</code>	125
Understanding Set User ID, Set Group ID, and Sticky Bit Permissions	125
Setting Permissions with Access Control Lists	127
Working with Files	128
Creating a File with <code>touch</code>	128
Creating a Directory with <code>mkdir</code>	129
Deleting a Directory with <code>rmdir</code>	129
Deleting a File or Directory with <code>rm</code>	130
Moving or Renaming a File with <code>mv</code>	131
Copying a File with <code>cp</code>	131
Displaying the Contents of a File with <code>cat</code>	132
Displaying the Contents of a File with <code>less</code>	132
Using Wildcards and Regular Expressions	133
Working as Root	133
Understanding and Fixing <code>sudo</code>	134
Creating Users	136
Deleting Users	137
Shutting Down the System	137
Rebooting the System	138
Commonly Used Commands and Programs	139
References	139

11 Command-Line Master Class, Part 1 141

Why Use the Command Line?	142
Using Basic Commands	143
Printing the Contents of a File with <code>cat</code>	144
Changing Directories with <code>cd</code>	145
Changing File Access Permissions with <code>chmod</code>	147
Copying Files with <code>cp</code>	147
Printing Disk Usage with <code>du</code>	148
Using <code>echo</code>	148
Finding Files by Searching with <code>find</code>	149
Searches for a String in Input with <code>grep</code>	151
Paging Through Output with <code>less</code>	152
Creating Links Between Files with <code>ln</code>	154

Finding Files from an Index with <code>locate</code>	156
Listing Files in the Current Directory with <code>ls</code>	156
Listing System Information with <code>lsblk</code> , <code>lshw</code> , <code>lsmod</code> , <code>lspci</code> , and <code>neofetch</code>	158
Reading Manual Pages with <code>man</code>	159
Making Directories with <code>mkdir</code>	160
Moving Files with <code>mv</code>	161
Renaming Files with <code>rename</code>	161
Deleting Files and Directories with <code>rm</code>	161
Sorting the Contents of a File with <code>sort</code>	162
Printing the Last Lines of a File with <code>tail</code>	163
Printing the Location of a Command with <code>which</code>	164
Downloading Files with <code>wget</code>	164
References	165
12 Command-Line Master Class, Part 2	167
Redirecting Output and Input	167
<code>stdin</code> , <code>stdout</code> , <code>stderr</code> , and Redirection	169
Comparing Files	170
Finding Differences in Files with <code>diff</code>	170
Finding Similarities in Files with <code>comm</code>	170
Limiting Resource Use and Job Control	171
Listing Processes with <code>ps</code>	171
Listing Jobs with <code>jobs</code>	173
Running One or More Tasks in the Background	173
Moving Jobs to the Background or Foreground with <code>bg</code> and <code>fg</code>	174
Printing Resource Usage with <code>top</code>	175
Setting Process Priority with <code>nice</code>	177
Combining Commands	178
Pipes	178
Combining Commands with Boolean Operators	180
Running Separate Commands in Sequence	180
Process Substitution	181
Executing Jobs in Parallel	181
Using Environment Variables	182
Using Common Text Editors	185
Working with <code>nano</code>	186
Working with <code>vi</code>	187
Working with <code>emacs</code>	188
Working with <code>sed</code> and <code>awk</code>	189
Working with Compressed Files	191
Using Multiple Terminals with <code>byobu</code>	192
Doing a Polite System Reset Using <code>REISUB</code>	194

Fixing an Ubuntu System That Will Not Boot	195
Checking BIOS	195
Checking GRUB	195
Reinstalling GRUB	195
Using Recovery Mode	196
Reinstalling Ubuntu	196
Tips and Tricks	196
Running the Previous Command	196
Running Any Previous Command	197
Running a Previous Command That Started with Specific Letters	197
Running the Same Thing You Just Ran with a Different First Word	197
Viewing Your History and More	197
Doing Two or More Things	198
Using Shortcuts	198
Confining a Script to a Directory	198
Using Coreutils	199
Reading the Contents of the Kernel Ring Buffer with <code>dmesg</code>	200
References	200
13 Managing Users	201
User Accounts	201
The Super User/Root User	202
User IDs and Group IDs	204
File Permissions	204
Managing Groups	205
Group Listing	205
Group Management Tools	206
Managing Users	207
User Management Tools	208
Adding New Users	209
Monitoring User Activity on the System	211
Managing Passwords	212
System Password Policy	212
The Password File	212
Shadow Passwords	214
Managing Password Security for Users	216
Changing Passwords in a Batch	216
Granting System Administrator Privileges to Regular Users	217
Temporarily Changing User Identity with the <code>su</code> Command	217
Granting Root Privileges on Occasion: The <code>sudo</code> Command	219

Disk Quotas	222
Implementing Quotas	222
Manually Configuring Quotas	223
Related Ubuntu Commands	223
References	224
14 Automating Tasks and Shell Scripting	225
What Is a Shell?	225
Scheduling Tasks	226
Using <code>at</code> and <code>batch</code> to Schedule Tasks for Later	227
Using <code>cron</code> to Run Jobs Repeatedly	229
Using <code>rtcwake</code> to Wake Your Computer from Sleep Automatically	231
Basic Shell Control	233
The Shell Command Line	233
Shell Pattern-Matching Support	235
Redirecting Input and Output	236
Piping Data	237
Background Processing	237
Writing and Executing a Shell Script	237
Running the New Shell Program	239
Storing Shell Scripts for System-wide Access	240
Interpreting Shell Scripts Through Specific Shells	240
Using Variables in Shell Scripts	242
Assigning a Value to a Variable	242
Accessing Variable Values	243
Positional Parameters	243
A Simple Example of a Positional Parameter	243
Using Positional Parameters to Access and Retrieve Variables from the Command Line	244
Using a Simple Script to Automate Tasks	244
Built-in Variables	246
Special Characters	247
Comparison of Expressions in <code>pdksh</code> and <code>bash</code>	250
Comparing Expressions with <code>tcsh</code>	255
The <code>for</code> Statement	259
The <code>while</code> Statement	261
The <code>until</code> Statement	263
The <code>repeat</code> Statement (<code>tcsh</code>)	263
The <code>select</code> Statement (<code>pdksh</code>)	264
The <code>shift</code> Statement	264
The <code>if</code> Statement	265

	The <code>expr</code> Statement	266
	The <code>case</code> Statement	267
	The <code>break</code> and <code>exit</code> Statements	269
	Using Functions in Shell Scripts	269
	References	270
15	The Boot Process	271
	Running Services at Boot	271
	Beginning the Boot Loading Process	272
	Loading the Linux Kernel	274
	Starting and Stopping Services with <code>systemd</code>	275
	Controlling Services at Boot with Administrative Tools	278
	Troubleshooting Runlevel Problems	278
	Boot-Repair	278
	References	279
16	System-Monitoring Tools	281
	Console-Based Monitoring	281
	Using the <code>kill</code> Command to Control Processes	283
	Using Priority Scheduling and Control	285
	Displaying Free and Used Memory with <code>free</code>	286
	Disk Space	286
	Disk Quotas	287
	Checking Log Files	287
	Rotating Log Files	289
	Graphical Process- and System-Management Tools	292
	System Monitor	292
	Conky	292
	Other Graphical Process- and System-Monitoring Tools	297
	KDE Process- and System-Monitoring Tools	298
	Enterprise Server Monitoring	298
	References	299
17	Backing Up	301
	Choosing a Backup Strategy	301
	Why Data Loss Occurs	302
	Assessing Your Backup Needs and Resources	303
	Evaluating Backup Strategies	304
	Making the Choice	308
	Choosing Backup Hardware and Media	308
	External Hard Drive	308
	Network Storage	308

Tape Drive Backups	309
Cloud Storage	309
Using Backup Software	309
tar: The Most Basic Backup Tool	310
The GNOME File Roller	312
The KDE ark Archiving Tool	312
Déjà Dup	313
Back In Time	314
Unison	315
Amanda	315
Alternative Backup Software	316
Copying Files	316
Copying Files Using tar	317
Compressing, Encrypting, and Sending tar Streams	318
Copying Files Using cp	318
Using rsync	319
Version Control for Configuration Files	320
System Rescue	323
The Ubuntu Rescue Disc	323
Restoring the GRUB2 Boot Loader	323
Saving Files from a Nonbooting Hard Drive	324
References	324
18 Networking	325
Laying the Foundation: The localhost Interface	326
Checking for the Availability of the Loopback Interface	326
Configuring the Loopback Interface Manually	327
Checking Connections with ping, traceroute, and mtr	328
Networking with TCP/IP	330
TCP/IP Addressing	331
Using IP Masquerading in Ubuntu	332
Ports	333
IPv6 Basics	334
Network Organization	337
Subnetting	337
Subnet Masks	337
Broadcast, Unicast, and Multicast Addressing	338
Hardware Devices for Networking	338
Network Interface Cards	338
Network Cable	340
Hubs and Switches	342
Routers and Bridges	343
Initializing New Network Hardware	343

Using Network Configuration Tools	345
Command-Line Network Interface Configuration	346
Network Configuration Files	350
Using Graphical Configuration Tools	355
Dynamic Host Configuration Protocol	355
How DHCP Works	356
Activating DHCP at Installation and Boot Time	357
DHCP Software Installation and Configuration	358
Using DHCP to Configure Network Hosts	359
Other Uses for DHCP	361
Wireless Networking	361
Support for Wireless Networking in Ubuntu	361
Choosing from Among Available Wireless Protocols	363
Beyond the Network and onto the Internet	363
Common Configuration Information	364
Configuring Digital Subscriber Line Access	365
Understanding PPP over Ethernet	366
Configuring a PPPoE Connection Manually	366
Configuring Dial-up Internet Access	367
Troubleshooting Connection Problems	368
References	369
19 Remote Access with SSH and VNC	371
Setting Up an SSH Server	371
SSH Tools	372
Using <code>scp</code> to Copy Individual Files Between Machines	372
Using <code>sftp</code> to Copy Many Files Between Machines	373
Using <code>ssh-keygen</code> to Enable Key-Based Logins	373
Virtual Network Computing	375
Guacamole	377
References	377
20 Securing Your Machines	379
Understanding Computer Attacks	379
Assessing Your Vulnerability	381
Protecting Your Machine	382
Securing a Wireless Network	382
Passwords and Physical Security	383
Configuring and Using Tripwire	384
Securing Devices	385
Viruses	385
Configuring Your Firewall	386

AppArmor	388
Forming a Disaster Recovery Plan	390
References	391
21 Performance Tuning	393
Storage Disk	394
Linux File Systems	394
The <code>hdparm</code> Command	395
File System Tuning	396
The <code>tune2fs</code> Command	396
The <code>e2fsck</code> Command	397
The <code>badblocks</code> Command	397
Disabling File Access Time	397
Kernel	398
Tuned	399
References	400
22 Kernel and Module Management	401
The Linux Kernel	402
The Linux Source Tree	403
Types of Kernels	405
Managing Modules	406
When to Recompile	408
Kernel Versions	409
Obtaining the Kernel Sources	409
Patching the Kernel	410
Compiling the Kernel	412
Using <code>xconfig</code> to Configure the Kernel	414
Creating an Initial RAM Disk Image	418
When Something Goes Wrong	418
Errors During Compile	418
Runtime Errors, Boot Loader Problems, and Kernel Oops	419
References	419
Part IV Ubuntu as a Server	
23 Sharing Files and Printers	421
Using Network File System	422
Installing and Starting or Stopping NFS	422
NFS Server Configuration	422
NFS Client Configuration	423

Putting Samba to Work	424
Manually Configuring Samba with <code>/etc/samba/smb.conf</code>	426
Testing Samba with the <code>testparm</code> Command	429
Starting, Stopping, and Restarting the <code>smbd</code> Daemon	429
Mounting Samba Shares	430
Network and Remote Printing with Ubuntu	431
Creating Network Printers	431
Using the CUPS GUI	433
Avoiding Printer Support Problems	434
References	435
24 Common Web Server Stacks	437
LAMP	437
LEMP	439
MEAN	440
References	441
25 Apache Web Server Management	443
About the Apache Web Server	443
Installing the Apache Server	444
Starting and Stopping Apache	444
Runtime Server Configuration Settings	446
Runtime Configuration Directives	446
Editing <code>apache2.conf</code>	447
Apache Multiprocessing Modules	449
Using <code>.htaccess</code> Configuration Files	450
File System Authentication and Access Control	452
Restricting Access with <code>Require</code>	452
Authentication	453
Final Words on Access Control	455
Apache Modules	455
<code>mod_access</code>	456
<code>mod_alias</code>	456
<code>mod_asis</code>	456
<code>mod_auth</code>	457
<code>mod_auth_anon</code>	457
<code>mod_auth_dbm</code>	457
<code>mod_auth_digest</code>	457
<code>mod_autoindex</code>	458
<code>mod_cgi</code>	458
<code>mod_dir</code> and <code>mod_env</code>	458
<code>mod_expires</code>	458

mod_headers.....	458
mod_include.....	459
mod_info and mod_log_config.....	459
mod_mime and mod_mime_magic.....	459
mod_negotiation.....	459
mod_proxy.....	459
mod_rewrite.....	459
mod_setenvif.....	460
mod_speling.....	460
mod_status.....	460
mod_ssl.....	460
mod_unique_id.....	460
mod_userdir.....	460
mod_usertrack.....	460
mod_vhost_alias.....	460
Virtual Hosting.....	461
Address-Based Virtual Hosts.....	461
Name-Based Virtual Hosts.....	461
Logging.....	463
HTTPS.....	464
References.....	466
26 Nginx Web Server Management.....	467
About the Nginx Web Server.....	467
Installing the Nginx Server.....	469
Installing from the Ubuntu Repositories.....	469
Building the Source Yourself.....	469
Configuring the Nginx Server.....	470
Virtual Hosting.....	473
Setting Up PHP.....	474
Adding and Configuring Modules.....	475
HTTPS.....	476
Reference.....	477
27 Other HTTP Servers.....	479
lighttpd.....	479
Yaws.....	480
Cherokee.....	480
Jetty.....	481
thttpd.....	481
Apache Tomcat.....	482

WildFly	482
Caddy	482
References	482
28 Administering Relational Database Services	483
A Brief Review of Database Basics	484
How Relational Databases Work	486
Understanding SQL Basics	487
Creating Tables	488
Inserting Data into Tables	489
Retrieving Data from a Database	490
Choosing a Database: MySQL Versus PostgreSQL	492
Speed	492
Data Locking	492
ACID Compliance in Transaction Processing to Protect Data Integrity	493
SQL Subqueries	494
Procedural Languages and Triggers	494
Configuring MySQL	494
Setting a Password for the MySQL Root User	495
Creating a Database in MySQL	496
Configuring PostgreSQL	498
Initializing the Data Directory in PostgreSQL	498
Creating a Database in PostgreSQL	499
Creating Database Users in PostgreSQL	499
Deleting Database Users in PostgreSQL	500
Granting and Revoking Privileges in PostgreSQL	500
Database Clients	501
SSH Access to a Database	501
Local GUI Client Access to a Database	503
Web Access to a Database	503
The MySQL Command-Line Client	504
The PostgreSQL Command-Line Client	505
Graphical Clients	506
References	507
29 NoSQL Databases	509
Key/Value Stores	512
Berkeley DB	512
Cassandra	513
etcd	513
Memcached and MemcacheDB	513

Redis	514
Riak	514
Scylla	514
Document Stores	514
CouchDB	515
MongoDB	516
BaseX	517
Wide Column Stores	517
BigTable	517
HBase	518
Graph Stores	518
Neo4j	518
OrientDB	519
HyperGraphDB	519
FlockDB	519
References	519
30 Virtualization on Ubuntu	521
KVM	523
VirtualBox	527
VMware	528
Xen	528
References	529
31 Containers and Ubuntu	531
LXC and LXD	532
Docker	533
Kubernetes	534
References	534
32 Ubuntu and Cloud Computing	535
Why a Cloud?	536
Software as a Service (SaaS)	537
Platform as a Service (PaaS)	537
Infrastructure as a Service (IaaS)	537
Metal as a Service (MaaS)	537
Things to Consider Before You Make Any Decisions	538
Ubuntu on the Public Cloud	538
OpenStack	538
Amazon Web Services	539
Google Cloud	539

Microsoft Azure	539
Hybrid Clouds	539
Canonical-Specific Cloud Offerings	540
Juju	540
Mojo: Continuous Delivery for Juju	541
Landscape	541
References	541
33 Managing Sets of Servers	543
Puppet	543
Chef	544
Ansible	544
SaltStack	544
CFEngine	545
Juju	545
Landscape	545
References	545
34 Handling Email	547
How Email Is Sent and Received	547
The Mail Transport Agent	548
Choosing an MTA	550
The Mail Delivery Agent	550
The Mail User Agent	551
Basic Postfix Configuration and Operation	552
Configuring Masquerading	554
Using Smart Hosts	554
Setting Message Delivery Intervals	555
Mail Relaying	555
Forwarding Email with Aliases	556
Using Fetchmail to Retrieve Mail	557
Installing Fetchmail	557
Configuring Fetchmail	557
Choosing a Mail Delivery Agent	560
Procmail	561
Spamassassin	561
Squirrelmail	561
Virus Scanners	562
Autoresponders	562
References	562

35	Proxying, Reverse Proxying, and Virtual Private Networks (VPNs)	563
	What Is a Proxy Server?	563
	Installing Squid	564
	Configuring Clients	564
	Access Control Lists	565
	Specifying Client IP Addresses	569
	Sample Configurations	570
	Virtual Private Networks (VPNs)	572
	Setting Up a VPN Client	573
	Setting Up a VPN Server	575
	References	577
36	Lightweight Directory Access Protocol (LDAP)	579
	Configuring the Server	580
	Creating Your Schema	580
	Populating Your Directory	582
	Configuring Clients	584
	LDAP Administration	584
	References	585
37	Name Serving with the Domain Name System (DNS)	587
	Understanding Domain Names	588
	DNS Servers	589
	DNS Records	590
	Setting Up a DNS Server with BIND	593
	References	595
Part V	Programming Linux	
38	Using Programming Tools	597
	Programming in C with Linux	598
	Using the C Programming Project Management Tools Provided with Ubuntu	599
	Building Programs with <code>make</code>	599
	Using the <code>autoconf</code> Utility to Configure Code	601
	Debugging Tools	602
	Using the GNU C Compiler	603
	Programming in Java with Linux	604
	Graphical Development Tools	605

IDEs and SDKs	605
Using the KDevelop Client	606
The Glade Client for Developing in GNOME	607
Beginning Mobile Development for Android	607
Hardware	608
Linux Kernel	608
Libraries	608
Android Runtime	608
Application Framework	608
Applications	609
Installing Android Studio	609
Creating Your First Android Application	609
Version Control Systems	609
Managing Software Projects with Git	610
Managing Software Projects with Bazaar	611
Managing Software Projects with Subversion	612
Continuous Integration and Continuous Delivery and DevOps Tools ...	613
CI/CD Tools	614
Chaos Engineering	614
Canonical-created Tools	614
Launchpad	614
Ubuntu Make	615
Creating Snap Packages	615
Bikeshed and Other Tools	616
References	618
39 Using Popular Programming Languages	621
Ada	622
Clojure	622
COBOL	623
D	624
Dart	624
Elixir	625
Elm	625
Erlang	625
Forth	626
Fortran	626
Go	626
Groovy	627
Haskell	627

Java	627
JavaScript	628
Kotlin	628
Lisp	629
Lua	629
Mono	629
OCaml	630
Perl	630
PHP	631
Python	631
Raku	631
Ruby	631
Rust	632
Scala	632
Scratch	632
Vala	633
References	633
40 Helping with Ubuntu Development	635
Introduction to Ubuntu Development	636
Setting Up Your Development System	637
Install Basic Packages and Configure	637
Creating a Launchpad Account	638
Setting Up Your Environment to Work with Launchpad	638
Fixing Bugs and Packaging	640
References	643
41 Helping with Ubuntu Testing and QA	645
Community Teams	645
Ubuntu Testing Team	646
Ubuntu QA Team	646
Bug Squad	647
References	647
Index	649

NOTE

Bonus Chapters 42–44 can be accessed online at [informit.com/title/9780136778851](https://www.informit.com/title/9780136778851).

Part VI Bonus Online Chapters

42 Using Perl	Web:1
Using Perl with Linux	Web:1
Perl Versions	Web:2
A Simple Perl Program	Web:2
Perl Variables and Data Structures	Web:4
Perl Variable Types	Web:4
Special Variables	Web:5
Perl Operators	Web:6
Comparison Operators	Web:6
Compound Operators	Web:7
Arithmetic Operators	Web:7
Other Operators	Web:8
Special String Constants	Web:8
Conditional Statements: <code>if/else</code> and <code>unless</code>	Web:9
<code>if</code>	Web:9
<code>unless</code>	Web:9
Looping	Web:10
<code>for</code>	Web:10
<code>foreach</code>	Web:10
<code>while</code>	Web:11
<code>until</code>	Web:11
<code>last</code> and <code>next</code>	Web:12
<code>do ... while</code> and <code>do ... until</code>	Web:12
Regular Expressions	Web:12
Access to the Shell	Web:13
Modules and CPAN	Web:14
Code Examples	Web:15
Sending Mail	Web:15
Purging Logs	Web:17
Posting to Usenet	Web:18
One-Liners	Web:19
Command-Line Processing	Web:19
References	Web:20
43 Using Python	Web:21
Python on Linux	Web:22
The Basics of Python	Web:23
Numbers	Web:23
More on Strings	Web:24
Lists	Web:27

Dictionaries	Web:30
Conditionals and Looping	Web:30
Functions	Web:33
Object Orientation	Web:34
Class and Object Variables	Web:34
Constructors and Destructors	Web:35
Class Inheritance	Web:36
The Standard Library and the Python Package Index	Web:38
References	Web:38
44 Using PHP	Web:39
Introduction to PHP	Web:39
Entering and Exiting PHP Mode	Web:40
Variables	Web:40
Arrays	Web:42
Constants	Web:43
References	Web:44
Comments	Web:44
Escape Sequences	Web:45
Variable Substitution	Web:46
Operators	Web:46
Conditional Statements	Web:48
Special Operators	Web:50
Switching	Web:51
Loops	Web:52
Including Other Files	Web:54
Basic Functions	Web:55
Strings	Web:55
Arrays	Web:58
Files	Web:60
Miscellaneous	Web:63
Handling HTML Forms	Web:66
Databases	Web:67
References	Web:69

About the Author

Matthew Helmke has used Ubuntu since 2005. He has written about Linux for several magazines and websites, is a lead author of *The Official Ubuntu Book*, and has coauthored both *A Practical Guide to Linux: Commands, Editors, and Shell Programming* and *The VMware Cookbook*. In his day job, Matthew works for Gremlin (<https://www.gremlin.com/>) making the Internet more reliable. Matthew first used UNIX in 1987, while studying Lisp on a Vax at the university. He has run a business using only free and open source software, has consulted, and teaches as an adjunct professor for the University of Arizona. You can find out more about Matthew at <https://www.linkedin.com/in/matthewhelmke/> or drop him a line with errata or suggestions at matthew@matthewhelmke.com.

Dedication

To Sage, Sedona, and Philip—the most amazing kids a guy could hope for; to Sandra and Evan, who are wonderful and welcome additions to our lives; to my grandfather for always believing in me and teaching me to believe in myself; and to my friends in the Linux, developer, sysadmin, cloud computing, DevOps, Site Reliability Engineering, and Chaos Engineering communities.

Acknowledgments

I want to thank the many people who helped with past editions, helpful comments and ideas, and technical edits and both formal and informal advice. I owe a huge debt of gratitude to the Ubuntu community, Canonical, and Mark Shuttleworth for inviting me to participate in the community years ago. Finally, thanks to my colleagues at Pearson, especially Debra Williams, for the trust placed in me and the opportunity to collaborate on projects like this one.

Register Your Book

Register your copy of *Ubuntu Linux Unleashed 2021 Edition* at informit.com for convenient access to downloads, updates, and corrections as they become available. To start the registration process, go to informit.com/register and log in or create an account*. Enter the product ISBN, 9780136778851, and click Submit.

*Be sure to check the box that you would like to hear from us to receive exclusive discounts on future editions of this product.

Figure Credits

Figures 1-1 to 1-9, 4-7 to 4-9, 8-1, 9-1 to 9-3, 11-1, 15-1, 16-1, 17-2, 19-1, 19-2, 35-1, 35-3, 36-1: Screenshots of Ubuntu © 2020 Canonical Ltd

Figures 4-1 to 4-4: Screenshots of GNOME © The GNOME Project

Figure 4-5: Screenshot of Kubuntu © 2020 Canonical Ltd

Figure 4-6: Screenshot of Xubuntu © 2020 Canonical Ltd

Figure 5-1: Screenshot of Mozilla Firefox ©1998–2020 by individual mozilla.org contributors.

Figure 5-2: Screenshot of Google Chrome ©2020 Google LLC.

Figure 8-2: Screenshot of Warsow © 2020, Chasseur de Bots

Figure 8-3: Screenshot of Scorched 3D © 2020 Scorched3D

Figure 8-4: Screenshot of Frozen Bubble © Guillaume Cottenceau

Figure 8-5: Screenshot of Supertux © Pelya

Figure 8-6: Screenshot of Battle of Wesnoth © 2003–2020 by The Battle for Wesnoth Project

Figure 8-7: Screenshot of Frets on Fire offers flashy graphics and familiar game play © Unreal Voodoo

Figure 8-8: Screenshot of FlightGear features stunning landscapes and technically accurate control panels © Free Software Foundation

Figure 8-9: Screenshot of In Speed Dreams © Speed Dreams

Figure 12-1: Screenshot of Command Prompt © Microsoft 2020

Figures 23-1, 23-2: Screenshots of CUPS © 2020 Apple Inc

Figure 30-1: Screenshot of Oracle VM VirtualBox Manager © 2020 Oracle

Cover image: Widegren/Karlsson/Photodisc/Getty Images

This page intentionally left blank

Introduction

We are pleased to present the 2021 edition of *Ubuntu Unleashed*. Ubuntu is a Linux-based computer operating system that has taken the world by storm. From its humble beginning in 2004, Ubuntu has risen to become a perennial favorite for desktop Linux, as well as a popular choice for servers.

Ubuntu descends from one of the oldest and most revered Linux distributions, Debian. Debian is assembled by a team of talented volunteers, is one of the most stable and customizable distributions of Linux, and is well respected for its quality and technological prowess. It is, however, an operating system for geeks; the bar for entry into the Debian realm is set high, and its user base tends to be highly proficient and expects new users to learn the ropes before joining in. That is both appropriate and okay.

Ubuntu has leveraged the quality of Debian to create an operating system that ordinary people can use. This doesn't mean that Ubuntu users are not technologically proficient, just that they do not have to be. In fact, many talented and respected software developers love Ubuntu because it enables them to concentrate on their specific interests instead of the details of the operating system. This book is for these people and for those who aspire to join their ranks.

If you are new to Linux, you have made a great decision by choosing this book. *Unleashed* books offer an in-depth look at their subjects, helping both beginner and advanced users move to a new level of knowledge and expertise. Ubuntu is a fast-changing distribution that has an updated release twice a year. We have tracked the development of Ubuntu from early on to make sure the information in this book mirrors closely the development of the distribution.

IN THIS INTRODUCTION

- ▶ Licensing
- ▶ Who This Book Is For
- ▶ What This Book Contains
- ▶ Conventions Used in This Book

A QUICK WORD ABOUT MARKETING

Almost all of the content in this book applies regardless of what Ubuntu release version you are using, as long as it is reasonably current. The book has been written to try to focus on information that is useful for the longest amount of time possible. Some chapters, like those covering installation or the basics of the default Ubuntu graphical user interface, will have their information change frequently. These chapters are the exception. The blurb on the cover of the book about which editions this book covers was added to account for these chapters and to denote clearly when the book was most recently revised.

Do not let the highly technical reputation of Linux discourage you, however. Many people who have heard of Linux think that it is found only on servers, used to look after websites and email. But that is far from the truth. Distributions like Ubuntu are making huge inroads into the desktop market. Corporations are realizing the benefits of running a stable and powerful operating system that is easy to maintain and easy to secure. The best part is that as Linux distributions are improved, the majority of those improvements are shared freely, allowing you to benefit from the additions and refinements made by one distribution, such as Red Hat, while continuing to use a different distribution, such as Ubuntu, which in turn shares its improvements. You can put Ubuntu to work today and be assured of a great user experience. Feel free to make as many copies of the software as you want; Ubuntu is freely and legally distributable all over the world, so no copyright lawyers are going to pound on your door.

Licensing

Software licensing is an important issue for all computer users and can entail moral, legal, and financial considerations. Many consumers think that purchasing a copy of a commercial or proprietary operating system, productivity application, utility, or game conveys ownership, but that is not true. In the majority of cases, the *end user license agreement (EULA)* included with a commercial software package states that you have paid only for the right to use the software according to specific terms. This generally means you may not examine, make copies, share, resell, or transfer ownership of the software package. More onerous software licenses enforce terms that preclude you from distributing or publishing comparative performance reviews of the software. Even more insidious licensing schemes (and supporting legislation, especially in the United States) contain provisions allowing onsite auditing of the software's use!

This is not the case with the open source software covered in this book. You are entirely free to make copies, share copies, and install the software on as many computers as you want. (We encourage you to purchase additional copies of this book to give as gifts, however.)

You can put your copy of Ubuntu to work right away in your home or at your place of business without worrying about software licensing, per-seat workstation or client licenses, software auditing, royalty payments, or any other type of payments to third parties. However, be aware that although much of the software included with Ubuntu is licensed

under the GPL, some packages are licensed under other terms. Those packages are subject to a variety of software licenses, and many of them fall under a broad definition known as *open source*. Some of these licenses include the Artistic License, the BSD License, the Mozilla Public License, and the Q Public License.

For additional information about the various GNU software licenses, browse to www.gnu.org. For a definition of open source and licensing guidelines, along with links to the terms of nearly three dozen open source licenses, browse to www.opensource.org.

Who This Book Is For

This book varies in coverage from deep to shallow over a wide range of topics. This is intentional. Some topics are Ubuntu specific and are not covered by any other book, and they deserve deep coverage here. There are some topics that every power user really must master. Other topics are things power users should know about so that they understand some history, know some other options, or simply have what they need to be able to listen and participate in further discussions with other technical people without being completely confused.

Some topics, like using the Linux command line, receive deep and extensive coverage because we believe that information to be vital to anyone who wants to be a power user or become a skilled DevOps guru. That topic gets three full chapters.

Other topics get only brief coverage so that people who are interested get a few guideposts to help them continue if they are interested. For example, Chapter 39, “Using Popular Programming Languages,” describes languages such as Ada and Fortran, along with more than 20 other programming languages. In that chapter, dozens of programming languages are covered in just a few pages. These are useful topics to some but not topics we would consider vital.

In addition, some topics are just too broad to be covered in great depth in this book, but they deserve mention because, again, an intermediate to advanced user should have at least a foundational knowledge of them. These are covered briefly and then information is provided to help you find more resources and expand your understanding, as needed.

Those Wanting to Become Intermediate or Advanced Users

Ubuntu Unleashed is intended for intermediate and advanced users or those who want to become intermediate and advanced users. Our goal is to give you a nudge in the right direction, to help you enter the higher stages by exposing you to as many different tools and ideas as possible; we give you some thoughts and methods to consider and spur you on to seek out more. Although the contents are aimed at intermediate to advanced users, new users who pay attention will benefit from the advice, tips, tricks, traps, and techniques presented in each chapter. Pointers to more detailed or related information are also provided at the end of each chapter.

If you are new to Linux, you might need to learn some new computer skills, such as how to research your computer’s hardware, how to partition a hard drive, and (occasionally) how to use a command line. This book helps you learn these skills and shows you how

to learn more about your computer, Linux, and the software included with Ubuntu. Most importantly, it helps you overcome your fear of the system by telling you more about what it is and how it works.

We would like to take a moment to introduce a concept called “The Three Levels of Listening” from Alistair Cockburn’s *Agile Software Development*. This concept describes how a person learns and masters a technique, passing through three levels:

- ▶ **Following**—The level where the learner looks for one very detailed process that works and sticks to it to accomplish a task.
- ▶ **Detaching**—The level where the learner feels comfortable with one method and begins to learn other ways to accomplish the same task.
- ▶ **Fluent**—The level where the learner has experience with or understanding of many methods and doesn’t think of any of them in particular while doing a task.

We all start at the first level and progress from there. Few reach the last level, but those who do are incredibly effective and efficient. People aiming for this level are the very ones for whom we intend this book.

Myriad books focus on the first set of users. This is not one of them. It is our goal in *Ubuntu Unleashed* to write just enough to get you from where you are to where you want or need to be. This is not a book for newcomers who want or need every step outlined in detail, although we do that occasionally. This is a book for people who want help learning about what can be done and a way to get started doing it. The Internet is an amazing reference tool, so this is not a comprehensive reference book. This book is a tool to help you see the landscape, to learn enough about what you seek to get you started in the right direction with a quality foundational understanding.

Sysadmins, Programmers, DevOps, and Site Reliability Engineering

System administrators, or sysadmins, are people who keep servers and networks up and running. Their role is sometimes called *operations*. They deal with software installation and configuration and security, and they do all the amazing things behind the scenes that let others use these systems for their work. They are often given less respect than they deserve, but the pay is good, and it is a ton of fun to wield the ultimate power over a computer system. It is also a great responsibility, and these amazing guys and gals work hard to make sure they do their jobs well, striving for incredible system uptime and availability. Ubuntu is an excellent operating system for servers and networks, and in this book you can find much of the knowledge needed to get started in this role.

Programmers are people who write software. They are sometimes called *developers*. Programmers work with others to create the applications that run on top of those systems. Ubuntu is a great platform for writing and testing software. This is true whether you are doing web application development or writing software for desktop or server systems. It also makes a great platform for learning new programming languages and trying out new ideas. This book can help you get started.

DevOps is a portmanteau of *developer* and *operations*. It signifies a blending of the two roles already described. The information technology (IT) world is changing, and roles are becoming less clear-cut and isolated from one another. In the past, it was common to witness battles between programmers excited about new technology and sysadmins in love with stability. DevOps realizes that neither goal is healthy in isolation but that seeking a balance between the two can yield great results by removing the barriers to communication and understanding that sometimes cause conflict within a team. Because of the rise of cloud computing and virtualization, which are also covered in this book, and more agile forms of development, DevOps is a useful perspective that enables people working in IT to do an even better job of serving their ultimate clients: end users. Site Reliability Engineering (SRE) began at a similar time as DevOps and from a distance looks almost the same. The major difference is that the SRE focus is solely on keeping sites and systems up and running, whereas DevOps has evolved a wider philosophy that may touch on people processes outside of the typical scope of SRE. This book is a great foundation for those who want to learn knowledge that will help with both roles and implementations, hopefully presented in a way that balances them nicely.

What This Book Contains

Ubuntu Unleashed is organized into six parts, described here:

- ▶ Part I, “Getting Started,” takes you through installing Ubuntu on your computer in the place of any other operating system you might be running, such as Windows.
- ▶ Part II, “Desktop Ubuntu,” is aimed at users who want to use Ubuntu on desktop systems.
- ▶ Part III, “System Administration,” covers both elementary and sophisticated details of setting up a system for specific tasks and maintaining that system.
- ▶ Part IV, “Ubuntu as a Server,” gives you the information you need to start building your own file, web, and other servers for use in your home or office.
- ▶ Part V, “Programming Linux,” provides a great introduction to how you can extend Ubuntu capabilities even further by using the development tools available.
- ▶ Part VI, “Bonus Chapters,” consists of three online chapters on using the Perl, Python, and PHP languages on Ubuntu. See informit.com/title/9780136778851.

Conventions Used in This Book

It is impossible to cover every option of every command included in Ubuntu. Besides, with the rise of the Internet and high-speed connections, reference materials are far less valuable today than they used to be because most of these details are only a quick Google search away. Instead, we focus on teaching you how to find information you need while giving a quality overview worthy of intermediate and advanced users. Sometimes this book offers tables of various options, commands, and keystrokes to help condense, organize, and present information about a variety of subjects.

To help you better understand code listing examples and sample command lines, several formatting techniques are used to show input and ownership. For example, if a command or code listing example shows input that you as the user would type, the input is formatted in boldface after the sample command prompt, as follows:

```
matthew@seymour:~$ ls
```

If typed input is required, as in response to a prompt, the sample typed input also is in boldface, like so:

```
Delete files? [Y/n] y
```

All statements, variables, and text that should appear on your display use the same boldface formatting. In addition, command lines that require root or super-user access are prefaced with the `sudo` command, as follows:

```
matthew@seymour:~$ sudo printtool &
```

Other formatting techniques include the use of italic for placeholders in computer command syntax, as shown here:

```
matthew@seymour:~$ ls directoryname
```

Computer terms or concepts are also italicized upon first introduction in text.

The following elements provide you with useful tidbits of information that relate to the discussion of the text:

NOTE

A note provides additional information you might find useful as you are working. A note augments a discussion with ancillary details or may point you to an article, a white paper, or another online reference for more information about a specific topic.

TIP

Tips contain special insights or time-saving techniques, as well as information about items of particular interest to you that you might not find elsewhere.

CAUTION

A caution warns you about pitfalls or problems before you run a command, edit a configuration file, or choose a setting when administering your system.

SIDEBARS CAN BE GOLDMINES

Just because something is in a sidebar does not mean that you will not find something new here. Be sure to watch for these elements, which bring in outside content that is an aside to the discussion in the text. In sidebars, you will read about other technologies, Linux-based hardware, and special procedures to make your system more robust and efficient.

Finally, you should know that all text, sample code, and screenshots in *Ubuntu Unleashed* were developed using Ubuntu and open source tools.

Read on to start learning about and using the latest version of Ubuntu.

This page intentionally left blank

CHAPTER 1

Installing Ubuntu and Post-Installation Configuration

Not that long ago, the mere mention of installing Linux struck fear into the hearts of mortal men. Thanks to a campaign of fear, uncertainty, and doubt (commonly referred to as FUD), Linux garnered a reputation as something of an elitist operating system that could be installed, configured, and used only by extreme computer geeks and professionals. Nowadays, it is a different story entirely, and Ubuntu is one of the easiest Linux distributions (*distros*) to install. This chapter covers how to get started with an install disc, including booting from an Ubuntu Live DVD or USB drive to test your system. The chapter then covers the actual installation of Ubuntu, looking at the various options available. The whole process is pain free with Ubuntu, as you are about to learn.

This chapter covers installation on a typical desktop or laptop computer system.

Before You Begin the Installation

Installing a new operating system is a major event, and you should make sure that you have properly thought through what is going to take place. The first thing to consider is how the hardware will be affected by the software that you propose to install. Although Ubuntu runs well on a variety of hardware, checking your hardware components is worthwhile because some bits of hardware do not work well with Ubuntu. This section provides some areas for you to investigate and think about; it might even save you hours of frustration when something goes wrong. Problems are becoming much less frequent, but they still crop up occasionally.

IN THIS CHAPTER

- ▶ Before You Begin the Installation
- ▶ Step-by-Step Installation
- ▶ Shutting Down
- ▶ Finding Programs and Files
- ▶ Software Updater
- ▶ The `sudo` Command
- ▶ Configuring Software Repositories
- ▶ System Settings
- ▶ Configuring Wireless Networks
- ▶ Troubleshooting Post-Installation Configuration Problems
- ▶ References

You start by researching and documenting your hardware. This information will prove helpful later on, during the installation.

Researching Your Hardware Specifications

At the absolute minimum, you should know the basics of your system, such as how much RAM you have installed and what type of mouse, keyboard, and monitor you have. Knowing the storage capacity and type of hard drive you have is important because it helps you plan how you will divide it for Ubuntu and troubleshoot if problems occur. A small detail, such as whether your mouse uses the USB or PS/2 interface, ensures proper pointer configuration—something that should happen without any problem, but you will be glad you know it in case something does go wrong. The more information you have, the better prepared you are for any problems.

You can make an inventory or at least a quick list of some basic features of your system. Again, the items you most want to know include the type and speed of your processor, the amount of installed memory, the size of your hard drive, the type of mouse, the capabilities of the display monitor (such as maximum resolution), and the number of installed network interfaces (if any).

DVD INSTALLATION JUMP START

To install Ubuntu Desktop from an install disc, you should first test whether your system is compatible by running Ubuntu from the DVD live. In general, a system must have at least a 2GHz dual-core processor, 25GB of hard drive space, 2GiB RAM, and a monitor with a display resolution of at least 1024×768. You also need a DVD drive or USB stick, memory card, or external drive for the installer media. Internet access is not required but is very helpful and strongly recommended. See <https://help.ubuntu.com/community/Installation/SystemRequirements> for a more detailed list of requirements.

Installation Options

Ubuntu is made available in two main forms: the Desktop DVD and the Server install DVD. (This list does not include derivative distributions like Kubuntu or Lubuntu or less commonly used methods such as the network install disk.) For most people, the Desktop DVD is what you want. The Server install DVD can get a LAMP (Linux, Apache, MySQL, and PHP) server up and running in about 20 minutes, but, as you learn in this book, all these components are available to the Ubuntu default distribution. An ISO image contains the entire contents of a CD or DVD in a single file that can be used as if it were a CD or DVD and that can be burned to a physical CD or DVD if desired; see Chapter 7, “Multimedia Applications,” to learn how to burn CDs and DVDs in Ubuntu. You can find a list of the currently available ISO images in a couple of places. The place that is best for most people to download from is www.ubuntu.com/download, which includes a nice graphical menu system and links to easy-to-read information and detailed instructions. Those with more specific requirements, such as a desire to use one of the official alternative Ubuntu versions, like Kubuntu or Lubuntu, can find what they need by navigating the menus at <https://cdimage.ubuntu.com>.

OFFICIAL UBUNTU FLAVORS

Ubuntu has several official variants, called *flavors*, as follows:

- ▶ Ubuntu
- ▶ Kubuntu
- ▶ Lubuntu
- ▶ Ubuntu Budgie
- ▶ Ubuntu Kylin
- ▶ Ubuntu Mate
- ▶ Ubuntu Studio
- ▶ Xubuntu

Almost everything in this book applies to any of these flavors. The exceptions include GUI-specific content, such as GNOME-specific descriptions, and content that refers to programs not installed by default, such as many of the server and programming options. For more information about Ubuntu flavors, see <https://wiki.ubuntu.com/UbuntuFlavors>.

To install via a USB thumb drive (one that holds at least 2GB), download the ISO image you need. You also need to download a program that enables you to use this image to create a bootable USB drive. For Windows, try Universal USB Installer (www.pendrivelinux.com/universal-usb-installer-easy-as-1-2-3/) or ISO Recorder (<https://isorecorder.alexfeinman.com/>) and follow the instructions given by the authors. For macOS, use Disk Utility to erase and format the USB drive and Etcher (www.balena.io/etcher/) to write. For Ubuntu, use the installed Startup Disk Creator program available in the default install. After the ISO is written to the USB drive, use it as you follow the installation instructions in this section. To install using a DVD that you create, download the ISO image you need from www.ubuntu.com/download or <https://cdimage.ubuntu.com>. You need to download a program to enable you to burn this image to physical media. For Windows, try either InfraRecorder (<https://infirarecorder.org>) or ISO Recorder (<https://isorecorder.alexfeinman.com/>) and follow the instructions given by the authors. For macOS, you can use Apple's Disk Utility, which is installed by default. For Ubuntu, right-click the icon for an ISO image and select Write to Disc. After the DVD is created, use it as you follow the installation instructions in this section.

UBUNTU ON MAC HARDWARE

There are sometimes problems installing Ubuntu on Mac hardware. Apple designs and configures its devices with the intent that only its own software should be run on its hardware. Be warned if you want to try to install Ubuntu or anything else on Mac hardware: Here be dragons. It can work, but be prepared to search the web and maybe ask questions in some of the places listed in Chapter 2, "Background Information and Resources." In addition, it is not currently possible to install Ubuntu or any Linux distro on modern Mac computers that have the Apple T2 storage controller because there is no Linux driver available for that hardware.

32-Bit Versus 64-Bit Ubuntu

Starting with Ubuntu 20.04 LTS, Ubuntu is only officially releasing a 64-bit version. However, we are keeping this section for the moment for those using an older release on older hardware. Nearly all consumer and enterprise hardware released in the last 15+ years is 64-bit, the exception being in specific categories using specific processors (certain embedded applications on ARM or Atom chips, for example), which are not the focus for pretty much anyone reading this book. If you have special needs like these, you may need to contact Canonical for ideas and support and may just be out of luck with Ubuntu.

All users who can use it are better off using the 64-bit version of Ubuntu. The main difference has to do with how a computer processor is able to register and use memory, but speed is also a factor. Here is why.

A computer with a 32-bit processor will be able to use a maximum of 4GB of memory (actually a bit less, but this is a more practical-focused book and not a computer science text, so the approximation is close enough for our uses; and yes, there are exceptions such as when using physical address extension (PAE), but this is a reasonable general rule for those just getting started). A computer with a 64-bit processor will be able to use up to a theoretical limit of 17 billion GB. More memory addresses means that you can store more data in RAM, which is much faster than processing it while reading from and writing to disks or other storage media.

If you are dealing with large amounts of data, this is a huge benefit. Processing audio or video, manipulating large databases of weather data, or playing 3D video games will be much smoother. It will also happen faster.

Speed is increased for another reason. When you have more memory addresses, it is kind of like when the supermarket has more checkout lines open. You can process more operations simultaneously. As a result of the extra capacity, variables and arrays in programs are processed more efficiently, function arguments are passed more easily, and even new data models are available for programmers and languages to use.

This requires some adaptation. Programs written for 32-bit operating systems must generally be adapted to take advantage of the capabilities of 64-bit processors. Although it is (usually, or at least often) possible to run 32-bit programs on a 64-bit processor, doing so is not always advantageous. However, in the Linux world, including Ubuntu, most software has been refactored and recompiled to take advantage of 64-bit processors and their capabilities. Software written for 64-bit processors is not backward compatible with 32-bit processors.

Early on, driver support and adaptation of software took time and was not complete enough to recommend using 64-bit Ubuntu by default. For about a decade, nearly all Intel- and AMD-based computer systems sold have been 64-bit, and the software has caught up. There is no reason to use anything else on this equipment.

If you browse to <https://ubuntu.com/releases>, you will find downloadable `.iso` files that will allow you to create a DVD from which you can boot/install Ubuntu.

If you are using a modern consumer- or enterprise-focused processor (for example, AMD Ryzen, Intel Core i3/5/7/9, or Xeon) you are all set.

Planning Partition Strategies

Partitioning is a topic that can make novice Linux users nervous. Coming from a Microsoft world, where you might be used to having just one hard drive, it can seem a bit strange to use an operating system that makes partitioning a hard drive possible or even preferable and common.

Depending on your needs, you can opt to have a single large partition to contain everything, which is the official recommendation of the Ubuntu community and developers. You might prefer to segment your installation across several partitions if you have advanced knowledge and specific needs.

If you are installing Ubuntu in a corporate or business environment, the needs of the business should be a primary concern. Be careful to ensure that you build in an adequate upgrade path that allows you to extend the life of the system and add any additional storage or memory.

Knowing how software is placed on your hard drive for Linux involves knowing how Ubuntu organizes its file system. This knowledge helps you make the most of hard drive space. In some instances, such as when you're planning to have user directories mounted via NFS or other means, this information can help head off data loss, increase security, and accommodate future needs. Create a great system, and you'll be the hero of information services. The Linux file system is covered along with commands to manipulate files and directories in Chapter 10, "Command-Line Beginner's Class."

To plan the best partitioning scheme, research and know the answers to these questions:

- ▶ How much disk space does your system require?
- ▶ Do you expect your disk space needs to grow significantly in the future?
- ▶ Will the system boot only Ubuntu, or do you need a dual-boot system?
- ▶ How much data requires backup, and what backup system will work best? (See Chapter 17, "Backing Up," for more information about backing up your system.)

The Boot Loader

If your system does not have a UEFI BIOS, then during installation, Ubuntu automatically installs *GRUB2* (*Grand Unified Boot Loader*) to the *Master Boot Record (MBR)* or to the *GPT (GUID Partition Table)* of your hard drive. Handily enough, it also detects any other operating systems, such as Windows, and adds entries in GRUB2 as appropriate. If you have a specific requirement not to install GRUB2 to the MBR, you need to install using the Alternate disc, which enables you to specify the install location for GRUB2.

DUAL BOOT NOT RECOMMENDED, BUT YOU CAN TRY IT IF YOU WANT

If you are attempting to create a dual-boot system using both Windows and Ubuntu—a system in which multiple operating systems exist on the hard drive and the user selects which one to use at boot time—it depends on your system as to whether this is easy or difficult. For systems that do not have a UEFI BIOS, you should install Windows first because it will overwrite the MBR and ignore any other operating systems on the disk. Ubuntu also overwrites the MBR, but it does so in a way that creates a boot menu that includes all operating systems it detects on the disk. If you do have a UEFI system, then GRUB2 is installed as an EFI application on the EFI system partition, which makes dual booting much less painful and more reliable.

Either way, you can get dual-booting to work, but in the past few years options have arisen that are better for most people.

If you decide you must dual boot, make sure you have your Windows recovery media available and that you either already have enough free space on your hard drive or know how to shrink the existing Windows partition and create a new partition on the hard drive for Ubuntu. No support or instructions for doing this are given in this book. If you need to use more than one operating system on the same hardware, this book recommends virtualization.

See Chapter 30, “Virtualization on Ubuntu,” for more information.

Installing from DVD or USB Drive

On most PCs, the BIOS supports booting directly from a CD, DVD, or USB drive and enables you to set a specific order of devices (such as hard drive, CD-ROM, or USB) to search for bootable software. Turn on your PC and set its BIOS if required (usually accessed by pressing a function key or the Del key after powering on) and then insert your Ubuntu install media and boot to install Ubuntu.

UEFI

If you have hardware that is from 2010 or newer, it probably includes a firmware interface called UEFI. The Unified Extensible Firmware Interface is a specification that defines how an operating system and the hardware interact. It replaces the BIOS mentioned earlier, although some manufacturers and most end users continue to use the term *BIOS*, just for consistency over time, even when they are using UEFI. In the past, UEFI has also been known to cause problems when you try to install a different operating system than the default one it came with on one of these machines. The 64-bit version of Ubuntu is typically quite reliable in supporting installation regardless. However, if you encounter difficulties, see <https://help.ubuntu.com/community/UEFI> for assistance.

Step-by-Step Installation

This section describes a basic step-by-step installation of Ubuntu from an install DVD or USB drive you create yourself using an ISO image you downloaded and wrote to a disk

or USB drive using the instructions provided earlier; just replace mentions of DVD with your install medium. The install process itself is fairly straightforward, and you should not encounter any real problems.

CAUTION

If you have anything at all on your computer that you want to save, back it up first. Installing an operating system has become easier to do, but it is still a major change. Your entire hard drive will be erased, and new information will be written to it. This is expected when installing Ubuntu to the entire hard drive, but it can even happen due to user error or gremlins (unexplained problems) when attempting a dual-boot installation. Back up anything you want to preserve. Save data and files to an external hard drive or other medium. You may even want to back up your entire operating system and current installation using something like Clonezilla (<http://clonezilla.org>). Whatever you do, go in with the perspective that everything currently on the computer will disappear. If this is okay, continue.

Having your computer connected to the Internet as you proceed is useful and recommended so that you can download updates while installing.

Installing

To get started, insert the install DVD into your drive and reboot your computer.

NOTICE

The installation process occasionally changes when new releases occur, but the overall idea is consistent. The screenshots you see here are probably accurate, but it is possible you may see a change or two. If you understand what you need to do, any changes should be trivial to you.

The initial screen offers a variety of languages for you to use during installation (see Figure 1.1) and two options. The Try Ubuntu option boots and runs Ubuntu from the DVD without making any changes to your system so that when you remove the DVD and reboot, everything will be as it was before. Install Ubuntu installs Ubuntu instead of your current operating system or alongside it (for dual-booting). Select Install Ubuntu to begin.

Figure 1.2 shows the Preparing to Install Ubuntu screen. If you select the two check boxes at the bottom, Ubuntu downloads extra software, such as multimedia codecs and any software updates that have been released since the disk was created, and includes them in the installation. Doing so is recommended.

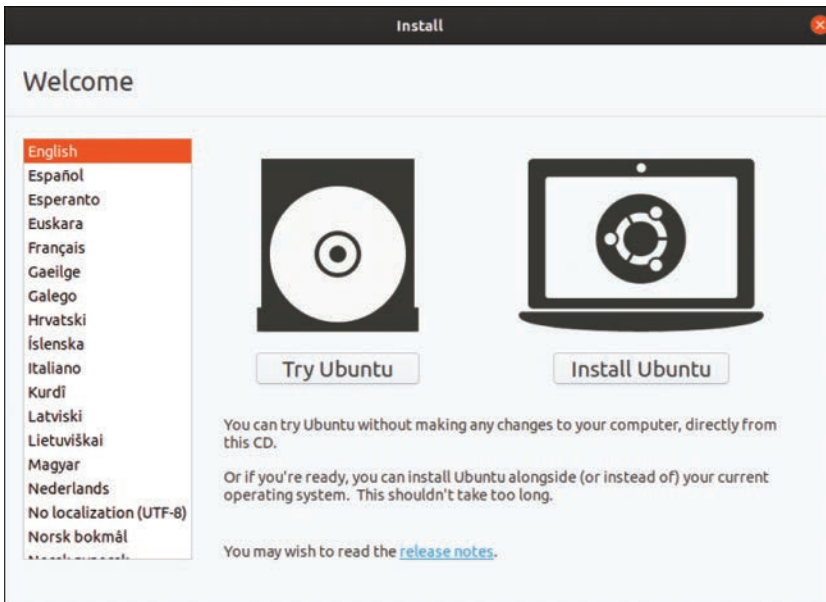


FIGURE 1.1 Choose a language for the installation in this opening screen.

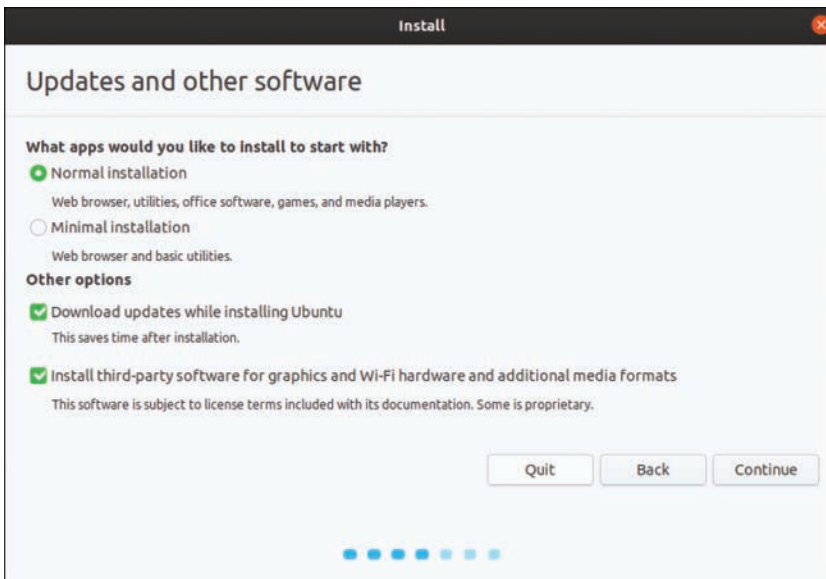


FIGURE 1.2 Before proceeding, decide whether you want to download updates during installation.

If other operating systems are found on your system, you are given the option to install Ubuntu alongside them or to erase them and use the whole disk for Ubuntu. See the dual-boot note and backup warning earlier in this chapter before continuing.

Next, as shown in Figure 1.3, you have the option either to erase and use the entire hard disk for your installation (recommended for people who are new to Linux and are nervous about partitioning a hard drive) or to specify partitions manually, which is not as difficult as it sounds. You also have the opportunity to have the entire Ubuntu installation encrypted; this is very secure, but if you lose your password, you are eternally out of luck, so choose wisely.

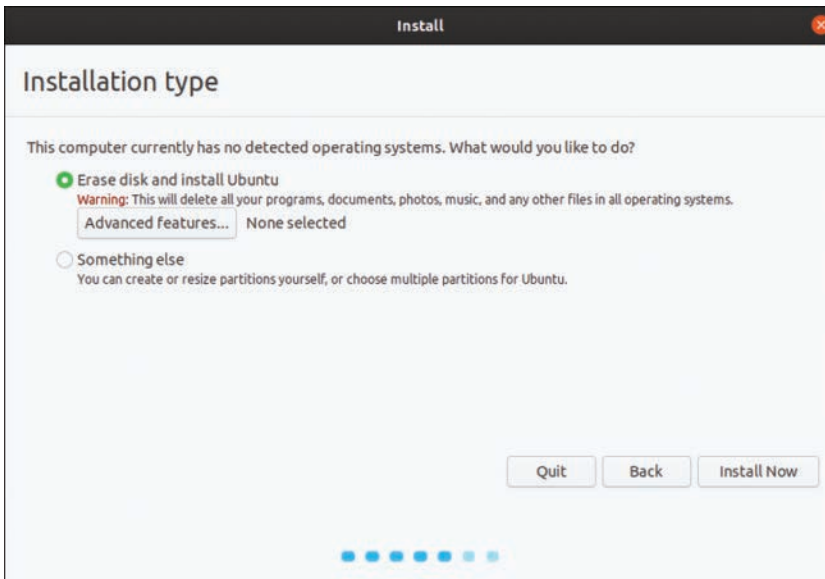


FIGURE 1.3 What do you want to do with your storage drive?

NOTE

More complex partitioning schemes might be useful for more complex systems, such as servers, especially when multiple hard drives are available. Read Chapter 10, “Command-Line Beginner’s Class,” for a quick discussion of the parts of the Linux file system to get an idea of parts of the file system that could benefit from residing on their own (or even multiple!) partitions or disks.

NOTE

Adding, formatting, and partitioning a hard drive can be done at any time, not only during installation. You should not try to modify a drive you are currently using, such as the one on which you have installed Ubuntu, because doing so could make the computer unusable, and you would have to start over. However, if you are adding a second drive or if you want to reformat a flash drive, partitioning the drive you’re using is a useful skill to know. In any case, this is an instance where the standard warning applies: Pay attention to what you are doing and know why you are doing it because this powerful tool both is useful and has the potential to cause you serious problems and data loss if used incorrectly.

The easiest and simplest method is to use GParted, which is a graphical partition manager. GParted is not installed by default, but it is available in the Ubuntu software repositories; see Chapter 9, “Managing Software,” if you need help installing it.

In addition to being useful for adding drives, GParted can also assist you in recovering from problems. You can run it after booting Ubuntu in a live session or running from a live CD or a USB drive. In this case, the system’s hard drives are not mounted, and you can manage them from the live session.

After you have made your partitioning selections, installation continues by asking about your current location; it helps you select the most appropriate keyboard layout for your system, based on that location and the most common language used there, and it asks you to enter your name, a username that you will use to log in to the system, and a password. You can even choose to encrypt your /home folder during the process.

A QUICK ASIDE ON PASSWORDS

When you create your password, be sure to remember what you entered. If you forget it, you cannot use your new system because you will not be able to log on to it.

When setting a password, the common advice is to make sure it has a mixture of letters and numbers to make it more secure. For instance, a good example of a historically recommended style of password is `T1a5c0p`. Although this might seem like garbage at first glance, you can remember it easily by thinking of the phrase *This Is A Good Choice Of Password*, shortened to *Tiagcop*, and substituting some of the letters with similar-looking numbers.

There are some reasons this might not be the best recommendation anymore because computer systems are much faster than they used to be. It is a true statement that the longer a password is, the harder it is to break. For this reason, the newest recommendation is to use a passphrase consisting of at least four words, perhaps something like *green monkeys chortle often*. There is no doubt that a password the length of four common words combined together would be harder to break than the `T1a5c0p` example. From that perspective, it seems like a no-brainer. On the other hand, a longer password that does not use any words found in a dictionary would be even better, but the problem here is that these passwords, and even the `T1a5c0p` example, can be hard to remember and may end up being written down on a sticky note next to the computer, perhaps even stuck to the monitor. That is worse, especially if you use good security and create a different password for every website and computer system that requires one.

One solution is to choose a really good password for your system, one that you will remember, like the four-word example or a long passphrase like *PeanutButterandJelly Sandwiches*, and then create one more good password to use with a password manager program like KeePassX (available in the Ubuntu software repositories; see Chapter 9, “Managing Software,” for details on how to find and install it), which can generate long, completely random passwords for you and keep them in a list that can only be viewed by accessing your system and then accessing the program, both of which will use good passwords. Let’s get back to the installation.

While you are answering the questions asked by the installer, the Ubuntu install begins to copy files to your hard drive. Performing these tasks in parallel makes the process even faster than it used to be. When you have input all necessary information and as the

installation completes, you see a series of information screens that are filled with interesting content about Ubuntu and are worth reading while you wait.

When the process is complete, you are prompted to restart the computer. Do so and remove the install media when it is ejected. Then log in when the reboot is complete. That's it.

In previous editions of this book, this chapter was longer and described a process that was sometimes confusing and fraught with peril. The Ubuntu developers deserve high praise for making the installation so incredibly easy and fast. The Linux kernel supports more hardware than ever before, and the Ubuntu kernel gurus (who make the decisions about what hardware modules to enable, among other things) do a great job, and most hardware works out of the box.

First Update

It used to be that the first thing you needed to do with your new system was update it to the latest package versions. You would do this mainly to ensure that you had the latest security updates available. Remember the first installation step, where we recommended checking the box to have software updates download during the installation process? If you did, the updates were acquired during the installation, and your system should already be up-to-the-minute current.

If you want to double-check that you have all the current versions of software and security updates installed, read Chapter 9 and Chapter 10 for more information.

Shutting Down

At some point, you are going to want to shut down your computer. As with most other things in Linux, there are different ways to do it. You can use the power icon located in the upper-right corner of your screen to access the power menu, shown in Figure 1.4.

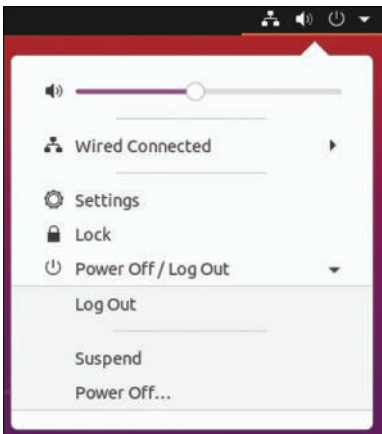


FIGURE 1.4 You can also lock the screen, access settings, and more from the power menu.

If you are working at the command line, you can immediately shut down your system by using the `shutdown` command, like this:

```
matthew@seymour:~$ sudo shutdown -h now
```

You can also use the `shutdown` command to restart your computer, as follows:

```
matthew@seymour:~$ sudo shutdown -r now
```

For new users, installing Ubuntu is the beginning of a new and highly rewarding journey on the path to learning Linux. For Ubuntu system administrators, the tasks ahead are to fine-tune the installation and to customize the server or user environment.

NOTE

Now that you have completed the primary task of installing Ubuntu, you can begin to customize your new operating system. This chapter looks at getting up and running with Ubuntu. Chapter 3, “Foundations of the Linux GUI,” gives you a tour around the desktop. Next, this chapter describes some basic administration tasks. By the end of this chapter, you should feel comfortable enough to move on through the rest of the book.

Finding Programs and Files

In the past, Ubuntu used a system of menus to guide users in searching for programs, but now there is an easier, faster way: Use search to find programs. You access search by clicking Activities at the upper left of the screen, and then start typing in the search box to find specific programs or documents on your system (see Figure 1.5). The desktop (also known as the GNOME user interface) is covered in more detail in Chapter 3.

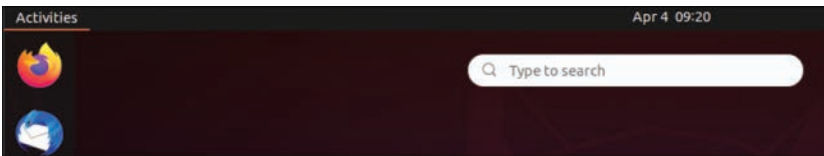


FIGURE 1.5 Search is the place to find programs and files.

Software Updater

Software Updater is covered in greater detail in Chapter 9, but it is worthy of a quick mention now so that from the start you can benefit from any available security and bug fixes. The easiest way to check for updates is to use Software Updater. Open Software Updater from search by typing **software updater** to search for it. When the window opens, Update Manager checks the Ubuntu software repositories to determine whether any updates are available. When it detects that new versions of installed software are available, it displays them in the window. Uncheck the check box next to any item if you don't

want to install that particular software update. Click Install Now to complete the process (see Figure 1.6). Software repositories are discussed later in this chapter.

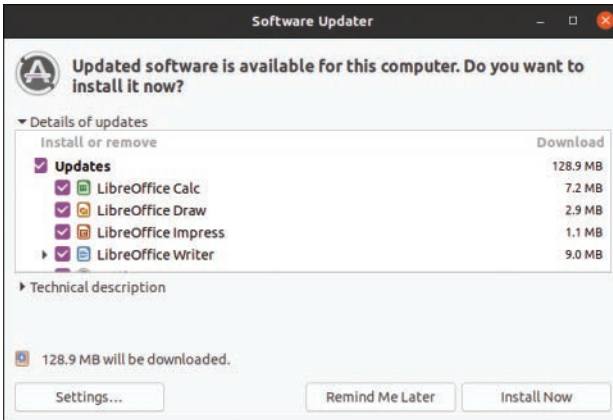


FIGURE 1.6 Software Updater, showing available software updates.

Another way of updating your system is to use the command line. This is vital on servers that do not have a GUI installed, and it is sometimes quicker than using Software Updater on a desktop computer. I like to use the command line to manage all the computers on my home network because I can use Secure Shell (SSH) to connect to each from a terminal and perform updates from another room in the house; anyone using that computer is left undisturbed while I'm making updates. You find out how to connect using the command line in Chapter 19, "Remote Access with SSH and VNC."

NOTE

In the second half of the "Software Updater" section and in "The `sudo` Command" section that follows, we introduce some commands that must be entered from the *command line*, also known as the *terminal*. Rather than a graphics-based user interface, the command line is a more traditional text-based user interface. Chapter 10, "Command-Line Beginner's Class," Chapter 11, "Command-Line Master Class, Part 1," and Chapter 12, "Command-Line Master Class, Part 2," cover this topic in much greater detail.

For now, the goal is to introduce you to the idea and let you know what can be done. You are not yet expected to know what a lot of this means.

To use the command line, open search and type `terminal`.

When you open the terminal, you are greeted with a prompt similar to the one here:

```
matthew@seymour:~$
```

A blinking cursor also displays. Ubuntu is awaiting your first command. Issue the following command:

```
matthew@seymour:~$ sudo apt update
```

This command tells the package management utility `apt` to check the Ubuntu repositories and look for any updates to your installed software. In a matter of seconds, Ubuntu completes all of this, and your screen should look something like this:

```
matthew@seymour:~$ sudo apt update
[sudo] password for matthew:
Hit:1 http://security.ubuntu.com bionic-security InRelease
Hit:2 http://us.archive.ubuntu.com bionic InRelease
Hit:3 http://us.archive.ubuntu.com/ubuntu bionic-updates InRelease
Hit:4 http://us.archive.ubuntu.com/ubuntu bionic-backports InRelease
Reading package lists... Done
Building dependency tree
Reading state information... Done
67 packages can be upgraded. Run 'apt list --upgradable' to see them.
matthew@seymour:~$
```

Upgrade your software by entering the following:

```
matthew@seymour:~$ sudo apt full-upgrade
```

Because you have already checked for updates, Ubuntu automatically knows to download and install only the packages it needs. The `full-upgrade` option works intelligently to ensure that any dependencies that are needed can be satisfied and will be installed, even if major changes are needed. You can also use the option `upgrade`, which isn't as smart as `full-upgrade`, but it might be a better choice on a production server because `upgrade` does not make major changes to software installations. It only makes changes that are necessary for security and simple package updates. This allows the systems administrator more flexibility to keep up to date with security while keeping running setups otherwise unchanged.

The `sudo` Command

You will find as you work through this book that Ubuntu relies on the `sudo` command while you work at the command line. This command is used in front of other commands to tell Ubuntu that you want to run the specified command with super user powers. This sounds really special, and it actually is. When you work using the `sudo` command, you can make wide-ranging changes to your system that affect the way it runs. Be extra careful when running any command prefixed with `sudo`, however; a wrong option or an incorrect command can have devastating consequences.

The use of `sudo` is straightforward. All you have to do is enter it like this:

```
matthew@seymour:~$ sudo command commandoptions
```

Just replace the word `command` with the command you want to run and `commandoptions` with any options. For example, the following command opens your `xorg.conf` file in `vi` and enables you to make any changes as the super user before being able to save it:

```
matthew@seymour:~$ sudo vi /etc/X11/xorg.conf
```

Whenever you execute a command using `sudo`, you are prompted for your password. This is the same password that you use to log in to Ubuntu, so it is important that you remember it.

Sometimes, however, you might want to work with a classic root prompt instead of having to type `sudo` in front of every command (if you have to work with lots of commands at the command line that require super user access, for example). `sudo` enables you to do this by using the `sudo -i` command. Again, you are prompted for your password, which you should enter, after which Ubuntu gives you the standard root prompt, as follows:

```
matthew@seymour:~#
```

From here, you can execute any command without having to keep entering `sudo`.

WARNING

Working from the root prompt can be really dangerous unless you know what you are doing. Until you are experienced, we recommend that you stick to using `sudo`, which is covered in more detail in Chapter 10.

Configuring Software Repositories

Ubuntu uses software repositories to get information about available software that can be installed on your system. Ubuntu is based on a much older Linux distribution called Debian. Debian has access to tens of thousands of different packages, which means that Ubuntu has access to these packages, too. The Debian packages are made available in Ubuntu's Universe repository. A set of volunteers called *Masters of the Universe (MOTUs)* are well trained and follow strict guidelines to package software and make even more packages available to Ubuntu users in the Universe repository. (See Chapter 40, "Helping with Ubuntu Development," for more information about the MOTUs and how you can become one of them.) The Universe repository is filled with optional and often useful or fun software; it is enabled by default, along with other official repositories containing security updates, software updates, and software necessary for Ubuntu to be installed and run in all of its various official forms.

You can adjust which repositories are enabled using the Software & Updates GUI tool, available in Software Updater by clicking Settings. On the first tab (Ubuntu Software), you have five options to choose from. The default settings are shown in Figure 1.7. Which options you check is entirely up to you, but make sure that at least the first check box is selected to allow you access to Canonical-supported open source software, which includes all the packages necessary for a basic Ubuntu installation and a few more that are commonly used. (Canonical is the company that funds much of Ubuntu development.) The more boxes you check, the wider your selection of software. Making sure that the Proprietary Drivers for Devices box is checked is also a good idea so that you can benefit from drivers that might enhance your system's performance.

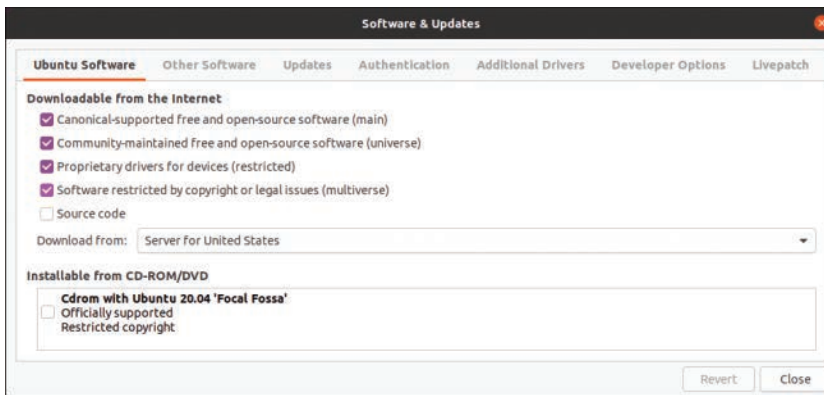


FIGURE 1.7 You can find or add other options under the Other Software, Updates, Additional Drivers, and other tabs.

OPEN SOURCE VERSUS PROPRIETARY

You might hear some arguments about using proprietary drivers or other software in Ubuntu. Some people feel that the use of such drivers goes against what *open source* stands for because the program code used for the drivers or software cannot be viewed and modified by the wider community but only by the original developers or company that owns it. There is also a strong argument that users should have to undergo the least amount of work for a fully functional system.

Ubuntu takes a middle-of-the-road stand on this and leaves it up to the user to decide. Open source software is installed by default, but options are given to allow proprietary software to be installed easily.

When you are happy with your selections, switch to the Updates tab to configure Ubuntu's behavior when updates are available (see Figure 1.8). By default, both the important security updates and recommended updates are checked to ensure that you have the latest bug fixes and patches. You can also choose to receive proposed updates and *backports* (software that is released for a newer version of Ubuntu but reprogrammed to be compatible with the current release), but we recommend this only if you are happy to carry out testing for the community because any updated software from these repositories can adversely affect your system.

Ubuntu also enables you to configure how often it checks for updates and how they are installed. By default, Ubuntu checks daily for updates and, if any are available, notifies you. However, you can change the frequency and the actions Ubuntu carries out when it finds available updates. We recommend keeping the notification-only option because this enables you to see what updates are available prior to their installation. If you want to save time, choose Download All Updates in the Background to configure Ubuntu to silently download the updates before it gives you the option to install them.

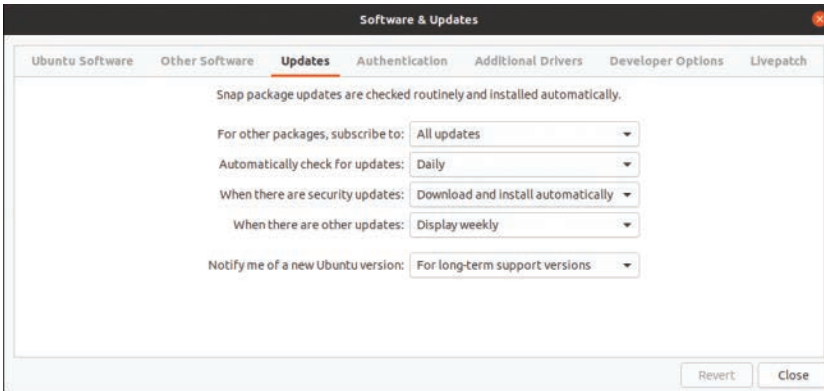


FIGURE 1.8 In the Updates tab of Software & Updates, configure which updates you want and how you want them to be handled.

Part of the magic of Ubuntu is the ease in which you can upgrade from major version to major version, such as moving from 19.04 to 19.10. Some Ubuntu releases are called LTS, for *long-term support*, and are intended for production use by most people. The interim releases are for those who feel they must have the most recent version of everything or those who help work on Ubuntu development. These releases are stable, but they are supported for only a short time, so if you choose to use them, you should plan to upgrade to the new release every six months. By ensuring that the release upgrade option is set to LTS releases only, you'll be prompted to upgrade your version of Ubuntu only every two years; the current LTS version used to write this book, 20.04, was released in April 2020, and the next LTS is scheduled to be released in April 2022.

The Other Software tab enables you to add other repositories. It comes by default with everything you need to connect to and use Canonical's partner repository, with nonfree (usually in the licensing sense, but occasionally for payment) software from companies that have an agreement with Canonical to make it easily available to interested users. This repository is disabled by default, and if you want to use it, you must enable it by checking a box next to its entry in the Other Software tab.

System Settings

To configure system settings, search for System Settings or click the settings logo in the power menu (refer to Figure 1.4). This opens the Settings window, shown in Figure 1.9, from which you can select entries from the menu at the left and then make adjustments as desired. A couple of the options are described in the following sections.

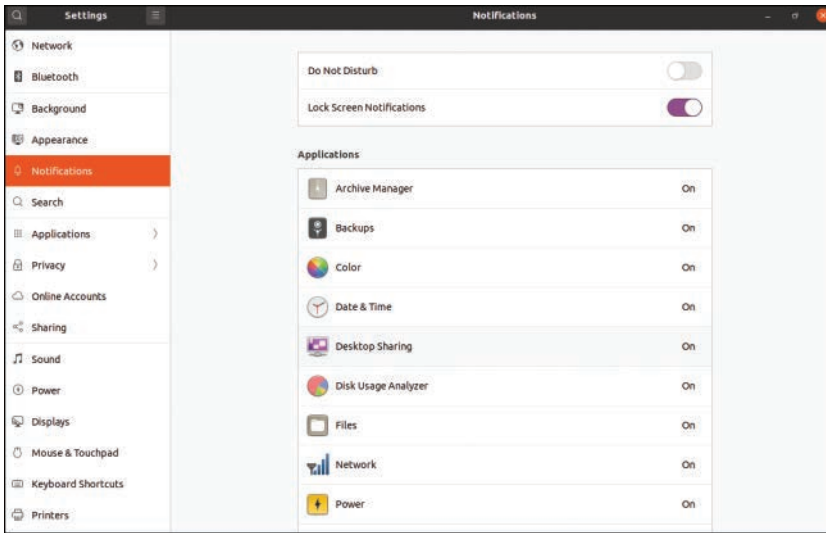


FIGURE 1.9 Adjust your system settings.

Detecting and Configuring a Printer

Setting up a printer in Linux used to be so difficult that previous editions of this book included an entire chapter filled with command-line magic and scary-looking configuration files. The setup is no longer that difficult in most cases.

Ubuntu includes drivers for many printers, and installing and using a printer in Ubuntu is usually easier than in other operating systems. (This is not an absolute rule, though.) Some printer manufacturers do not write and release drivers for Linux, and for some printers, no open source driver exists. Before you buy a printer, spending some time on the Internet searching for printers that are known to work with Linux is a good idea. One great resource is the Open Printing database from The Linux Foundation, at www.openprinting.org/printers.

If you choose wisely, all you need to do is plug your printer into the computer and turn it on. In many cases, Ubuntu finds the printer and adds the driver automatically. Within a couple of minutes, you should be able to use it. From the Settings window, select Devices and then select Printers to add a printer, to see all installed and configured printers, and to change printer settings. From here you can choose to enable printer sharing on a network, set options for default print quality, print a test page, and more.

Configuring Power Management in Ubuntu

Select Power from the Settings window to control how Ubuntu handles power-saving features in specific situations.

Ubuntu provides good support for *suspend*, which means your computer writes its current state to memory and goes into a low-power mode. Your computer will start much faster

the next time you use it because it does not need to perform a full boot; it brings the system up to its previous state out of memory instead of loading and starting every program again from scratch.

Setting the Time and Date

Linux provides a system time and date; your computer hardware provides a hardware clock-based time. In many cases, it is possible for the two times to drift apart. Linux system time is based on the number of seconds elapsed since January 1, 1970. Your computer's hardware time depends on the type of clock chips installed on your PC's motherboard, and many motherboard chipsets are notoriously subject to drift.

Keeping accurate time is important on a single workstation, but it is critically important in a network environment. Backups, scheduled downtimes, and other network-wide actions need to be accurately coordinated.

The Ubuntu installer sets the time and date during the installation process when it asks for your location. If you move or just want to change the settings (for example, to have your computer automatically synchronize its clock with official time servers on the Internet), you can do so.

Changing the Time and Date

Using Ubuntu's graphical tool is the simplest way to set your system date and time and the most obvious for a desktop user. From the Settings window, select Details and then select Date & Time to make changes. You can manually set the date and time in the GUI or have your computer obtain updated date and time information via the Internet.

Using the `date` Command

Use the `date` command to display or set your Linux system time. This command requires you to use a specific sequence of numbers to represent the desired date and time. To see your Linux system's idea of the current date and time, use the `date` command like this:

```
matthew@seymour:~$ date
Tue Sep 28 21:03:14 CDT 2021
```

To adjust your system's time (for example, to September 28, 2021 at 10:33 a.m.), use a command line with the month, day, hour, minute, and year, like so:

```
matthew@seymour:~$ sudo date 092810332021
Tue Sep 28 21:03:14 CDT 2021
```

Using the `hwclock` Command

Use the `hwclock` command to display or set your Linux system time, display or set your PC's hardware clock, or synchronize the system and hardware times. To see your hardware time and date, use `hwclock` with the `--show` option, like so:

```
matthew@seymour:~$ sudo hwclock --show
Tue 28 Sep 2021 06:04:43 PM CDT -0.281699 seconds
```

Use `hwclock` with its `--set` and `--date` options to manually set the hardware clock, as follows:

```
matthew@seymour:~$ sudo hwclock --set --date "09/28/21 10:33:00"
matthew@seymour:~$ hwclock --show
Tue 28 Sep 2021 10:33:09 AM MST -0.904668 seconds
```

In these examples, the hardware clock has been set using `hwclock`, which is then used again to verify the new hardware date and time. You can also use `hwclock` to set the Linux system time and date, using your hardware clock's values, with the Linux system time and date.

For example, to set the system time from your PC's hardware clock, use the `--hctosys` option, like so:

```
matthew@seymour:~$ sudo hwclock --hctosys
```

To set your hardware clock using the system time, use the `--systohc` option, like so:

```
matthew@seymour:~$ sudo hwclock --systohc
```

Configuring Wireless Networks

To manage networking from the GUI in Ubuntu, click the power icon (refer to Figure 1.4 earlier in this chapter). Entries in the menu that appears allow you to handle and monitor network connections.

Click the networking icon in the toolbar to connect to a wireless network. If your wireless access point broadcasts its *service set identifier (SSID)*, it should appear in the list under wireless networks. Click the desired network, and Network Manager detects what encryption (if any) is in use and asks you for the passkey. Enter this, and Network Manager starts the wireless connection. The passkey is then stored in the default keyring, a secure area that is unique to your login. From now on, whenever you log in to Ubuntu and are in range of this network, Network Manager will start the connection automatically.

If for some reason your wireless network does not appear (you might have your SSID hidden), you must use the Connect to Other Wireless Network option, in which case you enter the network name, wireless security type, and, when needed, the password for the connection.

Network Manager can also connect to Cisco VPN connections through use of the `vpnc` software. Install this from the Ubuntu repositories (see Chapter 9), and you can specify connection settings as appropriate, or if you have access to a predefined configuration file (PCF), you can import it directly into Network Manager.

Troubleshooting Post-Installation Configuration Problems

A lot of work has gone into making Ubuntu as versatile as possible, but sometimes you might come across a piece of hardware that Ubuntu is not sure about. Knowing what to do in these situations is important, especially when you are new to working with Ubuntu.

Because Ubuntu (and Linux in general) is built on a resilient UNIX foundation, it is much more stable than some other operating systems. However, even though things might seem to be working fine, Ubuntu could have a problem that might not affect the appearance of the system. In this section, you learn how to examine some of Ubuntu's built-in error logs, which can help you discover or diagnose unseen problems.

Ubuntu has a command that responds with detailed messages that are output directly by the operating system: the `dmesg` command, which we introduce here and cover more completely in Chapter 12. This command is commonly used with the `grep` command to filter output. The `dmesg` command takes its output directly from the `/var/log/syslog` file, so you can choose to either run `dmesg` or read the file directly by typing `less /var/log/syslog`. The output is fairly detailed, so be prepared for an initial shock when you see how much information is generated. You might find it easier to generate a file with the `dmesg` output by using the following command:

```
matthew@seymour:~$ dmesg > dmesg.txt
```

This takes the output from the `dmesg` command and stores it in a new text file called `dmesg.txt`. You can then browse it at your leisure, using your choice of text editor, such as `vi` or `emacs`. You can even use the `less` command, like so:

```
matthew@seymour:~$ less dmesg.txt
```

The messages are generated by the kernel, by other software run by `/etc/init.d`, and by Systemd scripts. You might find what appear at first glance to be errors, but some errors are not really problems (for example, if a piece of hardware is configured but not present on your system).

Thanks to Google, troubleshooting is no longer the slow process it used to be. You can copy and paste error messages into Google's search bar to bring up a whole selection of results similar to the problem you face. You are likely to come across people who have had the same problem as you.

It is important to work on finding and testing only one solution to one problem at a time; otherwise, you might end up getting no work done whatsoever. You should also get into the habit of making backup copies of all files that you modify, just in case you make a bad situation worse. Use the copy (`cp`) command like this:

```
matthew@seymour:~$ cp file file.backup20210101
```

You should not use a `.bak` extension on your backup files because this could get overwritten by another automatic process and leave you frustrated when you try to restore the original file. I like to use the convention `backupYYYYMMDD`, as in the preceding code, where I used the date for New Year's Day 2021: `2021(year)01(month)01(day)`.

If something breaks as a result of your changes to the original file, you can always copy the original back into place by using the command like this:

```
matthew@seymour:~$ cp file.backup20210101 file
```

NOTE

Something as simple as copying the original back into place can really save you, especially when you are under pressure because you've changed something you shouldn't have changed on a production system. The best practice is not to make sweeping changes on a production system.

If you are having trouble with booting your system, you may find it helpful to read Chapter 15, "The Boot Process," as it details aspects like the boot loader and startup process. Understanding what happens during this process will give you an idea of what to look for and how to solve problems.

References

- ▶ **www.ubuntu.com**—This is the place to start when looking for news, information, and documentation about installing, configuring, and using Ubuntu.
- ▶ **<https://ubuntu.com/download>**—This page has specific information and links for downloading, burning, and installing the current release of Ubuntu.
- ▶ **www.gnu.org/software/grub/**—This is the home page for the GRUB boot loader.

CHAPTER 2

Background Information and Resources

As with any new thing, finding out a bit about its history is worthwhile. Ubuntu is no different, and in this chapter you learn a little more about where Linux and Ubuntu came from. In addition, this chapter gives you resources to help you learn more on your own.

What Is Linux?

Linux is the core, or kernel, of a free operating system first developed and released to the world by Linus Benedict Torvalds in 1991. Torvalds, then a graduate student at the University of Helsinki, Finland, is now a Fellow at the Linux Foundation (www.linuxfoundation.org). He is an engineer and previously worked for the CPU design and fabrication company Transmeta, Inc., before leaving in 2003 to work for *Open Source Development Labs (ODSL)*, a consortium created by many high-tech companies to support Linux development, which has enabled him to focus on the Linux kernel full time. Fortunately for all Linux users, Torvalds chose to distribute Linux under a free software license called the GNU General Public License (GPL).

NOTE

The free online resource Wikipedia has a great biography of Linus Torvalds that examines his life and notable achievements. You can find it at https://en.wikipedia.org/wiki/Linus_Torvalds.

The GNU GPL is the brainchild of Richard M. Stallman, the founder of the Free Software Foundation. Stallman, the famous author of the `emacs` editing environment and GCC

IN THIS CHAPTER

- ▶ What Is Linux?
- ▶ Why Use Linux?
- ▶ What Is Ubuntu?
- ▶ Ubuntu for Business
- ▶ Ubuntu in Your Home
- ▶ Getting the Most from Linux and Ubuntu Documentation

compiler system, crafted the GPL to ensure that software that used the GPL for licensing would always be free and available in source-code form. The GPL is the guiding document for Linux and its ownership, distribution, and copyright. Torvalds holds the rights to the Linux trademark, but thanks to a combination of his generosity, the Internet, thousands of programmers around the world, GNU software, and the GNU GPL, Linux will remain forever free and unencumbered by licensing or royalty issues.

DISTRIBUTION VERSION AND KERNEL NUMBERING SCHEMA

There is a numbering system for Linux kernels, kernel development, and Ubuntu’s kernel versions. Note that these numbers bear no relation to the version number of your Ubuntu Linux distribution. Ubuntu distribution version numbers are assigned by the Ubuntu developers, whereas most of the Linux kernel version numbers are assigned by Linus Torvalds and his legion of kernel developers.

To see the date your Linux kernel was compiled, use the `uname` command with its `-v` command-line option. To see the version of your Linux kernel, use the `-r` option. The numbers, such as 4.14.0-22-generic, represent the major version (4), minor version (14), and patch level (0). The final number (22-generic) is the developer patch level and in our context is what is assigned by the Ubuntu developers.

Even minor numbers are considered “stable” and fit for use in production environments. You will find only stable versions of the Linux kernel included in this book. You can choose to download and install a beta (test) version of the kernel, but doing so is not recommended for a system destined for everyday use. Developers use beta kernels to test support of new hardware or operating system features.

Linux, pronounced “lih-nucks,” is free software. Combining the Linux kernel with GNU software tools—drivers, utilities, user interfaces, and other software such as the X.Org Foundation’s X Window System—creates a Linux distribution. There are many different Linux distributions from different vendors, but many derive from or closely mimic the Debian Linux distribution, on which Ubuntu is based.

NOTE

Debian lists several dozen other Linux distributions as being based on Debian Linux (see www.debian.org/misc/children-distros).

While it is really the kernel itself that is most appropriately referred to as “Linux,” colloquial language uses the term to refer to more than just the kernel. Most people who say they “use Linux” are referring to, at a minimum, a suite of software that includes several things. We have listed some of the more necessary ones here, in the order in which they are loaded into your computer’s memory during the boot cycle, after your computer’s BIOS or UEFI firmware (which was included by the manufacturer of the motherboard and which runs from where it is stored on the motherboard) has run to get things started:

- ▶ A boot loader, like GRUB2, which is described in Chapter 1, “Installing Ubuntu and Post-Installation Configuration”

- ▶ The Linux kernel, which is described in Chapter 22, “Kernel and Module Management”
- ▶ Daemons, which are background processes that the system runs to perform tasks like logging or listening for attempted network connections and so on; daemons may be more easily understood as programs that are not run or invoked directly by a user but that lie dormant until any of a specific set of conditions occurs
- ▶ The shell, which is a command processor that most people know best because it is what they see when they log in to the terminal; the shell is described in Chapter 14, “Automating Tasks and Shell Scripting”
- ▶ Shell utilities, such as most of the commands in Chapter 10, “Command-Line Beginner’s Class,” Chapter 11, “Command-Line Master Class, Part 1,” and Chapter 12, “Command-Line Master Class, Part 2”
- ▶ A graphical server, such as the X server, which is described in Chapter 3, “Foundations of the Linux GUI”
- ▶ A desktop environment, such as GNOME, which is also described in Chapter 3, and others such as those discussed in Chapter 4, “Ubuntu Desktop Options”
- ▶ Desktop software, such as web browsers, office suites, media players, games, and so on

A Linux distribution, like Ubuntu, collects all of these together, packages them, and makes them available to end users as a convenient set.

Why Use Linux?

Millions of clever computer users have been putting Linux to work for nearly 30 years. In recent years, many individuals, small office/home office (SOHO) users, businesses and corporations, colleges, nonprofits, and government agencies (local, state, and federal) in a number of countries have incorporated Linux with great success. And, today, Linux is being incorporated into many *information service/information technology (IS/IT)* environments as part of improvements in efficiency, security, and cost savings. Using Linux is a good idea for a number of reasons, including the following:

- ▶ **Linux provides an excellent return on investment (ROI)**—There is little or no cost on a per-seat basis. Unlike commercial operating systems, Linux has no royalty or licensing fees, and a single Linux distribution on a CD-ROM or in a network shared folder can form the basis of an enterprise-wide software distribution, replete with applications and productivity software. Custom corporate CD-ROMs can be easily crafted, or network shares can be created to provide specific installs on enterprise-wide hardware. This feature alone can save hundreds of thousands, if not millions, of dollars in IS/IT costs—all without the threat of a software audit from the commercial software monopoly or the need for licensing accounting and controls of base operating system installations.

- ▶ **Linux can be put to work on the desktop**—Linux, in conjunction with its supporting graphical networking protocol and interface (the X Window System), has worked well as a consumer UNIX-like desktop operating system since the mid-1990s. The fact that UNIX is ready for the consumer desktop is clear as Apple’s macOS is a POSIX-compliant operating system built on top of the Apple-developed XNU kernel. XNU stands for “X is Not Unix,” which is amusing because Apple eventually sought and achieved UNIX 03 certification, making macOS (then Mac OS X) an official version of UNIX.
- ▶ **Linux can be put to work as a server platform**—Linux is fast, secure, stable, scalable, and robust. The latest versions of the Linux kernel easily support multiple-processor computers, large amounts of system memory, individual file sizes in excess of hundreds of gigabytes, a choice of modern journaling file systems, hundreds of process-monitoring and control utilities, and the (theoretical) capability to simultaneously support more than 4 billion users. IBM, Oracle, and other major database vendors all have versions of their enterprise software available for Linux.
- ▶ **Linux has a low entry-and-deployment cost barrier**—Maintenance costs can also be reduced because Linux works well on a variety of PCs. Although the best program performance will be realized with newer hardware, base installs can even be performed on lower-end computers or embedded devices. This provides for a much wider user base, extends the life of older working hardware, and can help save money for home, small business, and corporate users.
- ▶ **Linux appeals to a wide audience in the hardware and software industry**—Versions of Linux exist for nearly every CPU. Embedded-systems developers now turn to Linux when crafting custom solutions using ARM, MIPS, and other low-power processors on platforms such as Raspberry Pi. Linux is also available for Intel’s Itanium CPU, as well as the AMD64 group of CPUs.
- ▶ **Linux provides a royalty-free development platform for cross-platform development**—Because of the open source development model and availability of free, high-quality development tools, Linux provides a low-cost entry point to budding developers and tech industry startups.
- ▶ **Big-player support in the computer hardware industry from such titans as IBM now lends credibility to Linux as a viable platform**—IBM has enabled Linux on the company’s entire line of computers, from low-end laptops to “big iron” mainframes. New corporate customers are using Linux as part of enterprise-level computing solutions. It has been used on some of the world’s fastest computers. Companies like HP and Dell also certify Linux across a large portion of their hardware offerings.

Look forward to even more support as Linux usage spreads worldwide throughout all levels of businesses in search of lower costs, better performance, and stable and secure implementations.

What Is Ubuntu?

Ubuntu is an operating system based on the Linux kernel, created, improved, refined, and distributed by the Ubuntu Community at www.ubuntu.com. Sponsored by Canonical Ltd. (www.canonical.com), Ubuntu is an open source project supported by a worldwide community of software developers.

Ubuntu released its first version in October 2004. It quickly gained a reputation for ease of installation and use, combined with the slightly wacky code names given to each release. However, Ubuntu itself is based on Debian, which is a much older distribution, well-respected by the wider Linux community. Debian is the rock on which Ubuntu is founded.

Sponsored by Canonical Software and with the formidable resources of Mark Shuttleworth, Ubuntu got off to a great start with version 4.10, the Warty Warthog. From the start, Ubuntu specified clear goals: to provide a distribution that was easy to install and use, that did not overly confuse the user, and that came on a single CD (now one DVD image). Releasing every six months, Ubuntu made rapid progress into the Linux community and is now one of the most popular Linux distros in the world.

UBUNTU VERSION NUMBERS

As mentioned earlier, Ubuntu has chosen a unique numbering scheme and some peculiar code names for releases since the first launch in October 2004. Doing away with the typical version numbering found elsewhere, Ubuntu decided to take the month and year of release and reverse them. Hence, the first release in October 2004 became 4.10, followed quickly by 5.04 (April 2005), 5.10, 6.06LTS, and so on up to the current 20.04.

The version covered in this book was released in April 2020 and therefore bears the version number 20.04. What's even more special about some releases is that they also carry the *LTS* (*long-term support*): Canonical will support LTS versions for three years on the desktop version and a total of five years for the server version after its release. LTS releases come out every two years, and the most recent LTS version is 20.04.

The code names during development are even better: 4.10 was christened the Warty Warthog, in recognition of the fact that it was a first release, warts and all. The second release, 5.04, was dubbed the Hoary Hedgehog. Things got slightly better with 5.10, code-named the Breezy Badger. 6.06 was announced as the Dapper Drake and was the first Ubuntu distribution to carry the LTS badge. Beyond Dapper, there was the Edgy Eft (6.10) followed by the Feisty Fawn (7.04), and more. For a full list of development code names, see <https://wiki.ubuntu.com/DevelopmentCodeNames>.

Ubuntu for Business

Linux has matured over the years. It includes all the essential features for use in enterprise-level environments, such as CPU architecture support, file systems, and memory handling. Linux provides the foundation for cloud computing, containers, and even Android.

Small business owners can earn great rewards by stepping off the software licensing and upgrade treadmill and adopting a Linux-based solution. A business using Ubuntu not only avoids the need for licensing accounting and the threat of software audits but also has viable alternatives to many types of commercial productivity software, often for free.

Using Ubuntu in a small business setting makes a lot of sense for other reasons, too, such as not having to invest in cutting-edge hardware to set up a productive shop. Ubuntu easily supports older, or *legacy*, hardware, and savings are compounded over time by avoiding unnecessary hardware upgrades. Additional savings will be realized because OS software and upgrades are free. New versions of applications can be downloaded and installed at little or no cost, and office suite software is free.

Ubuntu is easy to install on a network and plays well with others, meaning it works well in a mixed-computing situation with other operating systems such as Windows, macOS, and of course, UNIX. A simple Ubuntu server can be put to work as an initial partial solution or made to mimic file, mail, or print servers of other operating systems. Clerical staff should quickly adapt to using familiar Internet and productivity tools, while your business gets the additional benefits of stability, security, and a virus-free computing platform.

By carefully allocating monies spent on server hardware, a productive and efficient multiuser system can be built for much less than the cost of comparable commercial software. Combine these benefits with support for laptops, mobile devices, and remote access, and you will find that Ubuntu supports the creation and use of an inexpensive yet efficient work environment.

Ubuntu in Your Home

Ubuntu installs a special set of preselected software packages onto your hard drive. These packages, which are suitable for small office/home office (SOHO) users, provide a wealth of productivity tools for document management, printing, communication, and personal productivity.

The standard installation requires only a small amount of drive space. Despite the small footprint, the install also contains administrative tools, additional authoring and publishing clients, a variety of editors, a lovely desktop, and much more.

Getting the Most from Linux and Ubuntu Documentation

You can find links to Ubuntu documentation at www.ubuntu.com.

You will find traditional Linux software package documentation, such as manual pages, under the `/usr/share/man` directory, with documentation for each installed software package under `/usr/share/doc`.

Linux

Linux manual pages are compressed text files that contain succinct information about how to use a program. Each manual page generally provides a short summary of a command's use, a synopsis of command-line options, an explanation of the command's purpose, potential caveats or bugs, the name of the author, and a list of related configuration files and programs.

For example, you can learn how to read manual pages by using the `man` command to display its own manual page, as follows:

```
matthew@seymour:~$ man
```

After you press Enter, a page of text appears on the screen or in your window on the desktop. You can scroll through the information by using your keyboard's cursor keys, read, and then press the Q key to quit reading.

Many of the software packages also include separate documents known as HOWTOs that contain information regarding specific subjects or software. With a HOWTO document that is a simple text file in compressed form (with a filename ending in `.gz`), you can easily read the document by using the `zless` command, which is a text pager that enables you to scroll back and forth through documents. (Use the `less` command to read plain-text files.) You can start the command by using `less`, followed by the complete directory specification and name of the file, or *pathname*, like this:

```
matthew@seymour:~$ less /usr/share/doc/httpd-2.0.50/README
```

To read a compressed version of this file, use the `zless` command in the same way:

```
matthew@seymour:~$ zless /usr/share/doc/attr-2.4.1/CHANGES.gz
```

After you press Enter, you can scroll through the document using your cursor keys. Press the Q key to quit.

With a HOWTO document in HTML format, you can simply read the information using a web browser, such as Firefox. If you are reading from a console, you can use the links or Lynx text-only web browsers, like this:

```
matthew@seymour:~$ links /usr/share/doc/stunnel-4.0.5/stunnel.html
```

The links browser offers drop-down menus, accessed by clicking at the top of the screen. You can also press the Q key to quit.

With documentation in PostScript format (with filenames ending in `.ps`), you can use the `gv` client to read or view the document, like this:

```
matthew@seymour:~$ gv /usr/share/doc/iproute-2.4.7/ip-crefs.ps
```

Finally, if you want to read a document in Portable Document Format (with a filename ending in `.pdf`), use the `evince` client, as follows:

```
matthew@seymour:~$ evince /usr/share/doc/xfig/xfig-howto.pdf
```

NOTE

This book was developed and written using software from Ubuntu. You can download your own copy, available as ISO9660 images (with filenames ending in `.iso`), and burn it onto a DVD or create a bootable USB stick.

Along with the full distribution, you get access to the complete source code for the Linux kernel and source for all software in the distribution—more than 55 million lines of C and nearly 5 million lines of C++ code. Browse to www.ubuntu.com/download/ to get started.

Ubuntu

The best place to start for Ubuntu-specific information is at Ubuntu-focused websites. Where better to start than the main website for the distribution and the official web forums? Although these are not the only official Ubuntu resources, they are likely to be immediately useful. You can easily find others under the Support tab on the Ubuntu.com website:

- ▶ **www.ubuntu.com**—This is the home page for Ubuntu, Canonical’s community-based free Linux distribution. Ubuntu is the main release of this Linux distribution and includes thousands of software packages that form the core of an up-to-date, cutting-edge Linux-based desktop. You can also find links to the other *buntus, such as Kubuntu, Xubuntu, and EdUbuntu.
- ▶ **<https://help.ubuntu.com>**—This is the place to start for official Ubuntu documentation.
- ▶ **www.ubuntuforums.org**—This is a good place to go if you need specific community-provided Ubuntu support.
- ▶ **<https://askubuntu.com>**—This is another good place to go if you need specific community-provided Ubuntu support.
- ▶ **<https://tutorials.ubuntu.com>**—This is a newer resource with Canonical-provided guides to performing specific tasks.
- ▶ **<https://community.ubuntu.com>**—This is a light-traffic site designed to help coordinate work in the Ubuntu community.
- ▶ **<https://answers.launchpad.net/ubuntu>**—This is the official bug reporting system and tracker for Ubuntu.

CHAPTER 3

Foundations of the Linux GUI

Imagine a world of black screens with white text or dark green screens with light green text. You may or may not remember it, but that used to be the primary interface for users to access computers. Computing has moved on significantly and has adopted the *graphical user interface (GUI)* as standard on most desktop and workstation platforms. Not only that, but GUIs have gradually changed and evolved over time. This chapter covers low-level information about what lies underneath the GUI, whereas Chapter 4, “Ubuntu Desktop Options,” covers desktop software options, starting with the default, GNOME.

Foundations and the X Server

Many Linux distributions are switching to Wayland (see the nearby sidebar). For 20.04 LTS, however, Ubuntu continues to use the X Window System, the classic and stable graphical networking interface that provides the foundation for a wide range of graphical tools and window managers. More commonly known as just X, it can also be referred to as X11R7 and X11. Coming from the world-renowned Massachusetts Institute of Technology, X has gone through several versions, each of which has extended and enhanced the technology. The open source implementation is managed by the X.Org foundation, whose board includes several key figures from the open source world. On September 15, 2017, X11 turned 30 years old, which is a very long time for any software to remain usable and be in active development. Ubuntu will eventually move away from X to Wayland by default, but that is not yet the case as of the release of Ubuntu 20.04 LTS, and many third-party applications are not yet fully adapted to work as well on Wayland as they

IN THIS CHAPTER

- ▶ Foundations and the X Server
- ▶ Starting X
- ▶ References

do on X. Wayland is available to users as an option on the login screen, should you prefer to give it a try.

WHAT IS WAYLAND?

The short answer is that Wayland is intended as a simpler window manager replacement for X.

The more complex answer is that Wayland is a compositing window manager protocol that defines how to communicate between the compositor in the Wayland window manager server and its application window clients. It also includes a C library implementation of that protocol. A compositor is an application that provides a separate region of physical memory storage for each GUI window (this memory location is called a buffer). Each frame of each running application uses a separate memory location, and the window manager then composites them together. When done well, this provides a smoother desktop experience with little or no flickering, along with the ability to perform additional processing on buffered windows, such as flashy effects. Compositing is different from how X works, as X builds bitmaps to create the graphical user interface, where each item on the screen corresponds to one or more bits of information, most often defining a display place and color for each pixel.

To learn more about Wayland, see <https://wayland.freedesktop.org> and <https://wiki.ubuntu.com/Wayland> for Ubuntu-specific plans and details.

The best way to think about how X works is to see it as a client-server system. The X server provides services to programs that have been developed to make the most of the graphical and networking capabilities that are available under the server and in the supported libraries. X.Org provides versions for many platforms, including Linux and macOS.

A desktop environment for X provides one or more window managers and a suite of clients that conform to a standard graphical interface based on a common set of software libraries. When used to develop associated clients, these libraries provide graphical consistency for the client windows, menus, buttons, and other onscreen components, along with some common keyboard controls and client dialogs.

Basic X Concepts

The underlying engine of X11 is the X protocol, which provides a system of managing displays on local and remote desktops. The protocol uses a client-server model that allows an abstraction of the drawing of client windows and other decorations locally and over a network. An X server draws client windows, dialog boxes, and buttons that are specific to the local hardware and in response to client requests. The client, however, does not have to be specific to the local hardware. This means that system administrators can set up a network with a large server and clients and enable users to view and use those clients on workstations with totally different CPUs and graphics displays.

Because X offers users a form of distributed processing, Ubuntu can be used as a very cheap desktop platform for clients that connect to a powerful X server. The more powerful the X server, the larger the number of X-based clients that can be accommodated. This functionality can breathe new life into older hardware, pushing most of the graphical

processing on to the server. A fast network is a must if you intend to run many X clients, because X can become bandwidth hungry.

X is hugely popular in the UNIX and Linux world for a variety of reasons. That it supports nearly every hardware graphics system is a strong point. This and strong multiplatform programming standards give it a solid foundation of developers committed to X. Another key benefit of X is its networking capability, which plays a central role in administration of many desktops and can also assist in the deployment of a thin-client computing environment. Being able to launch applications on remote desktops and being able to standardize installations are examples of the versatility of this powerful application.

More recent versions of X have also included support for shaped windows (that is, non-rectangular), graphical login managers (also known as *display managers*), and compressed fonts. Each release of X brings more features designed to enhance the user experience, including being able to customize how X client applications appear, right down to buttons and windows. Most office and home environments run Linux and X on their local machines. Enlightened companies and users harness the power of the networking features of X, enabling thin-client environments and allowing the use of customized desktops designed specifically for them. Having applications launch from a single location makes the lives of system administrators a lot easier because they have to work on only one machine rather than several.

Using X

X.Org (www.x.org) is the X server that is used with Ubuntu. The base distribution consists of many packages, including the server, support and development libraries, fonts, various clients, and documentation. An additional 1,000 or more X clients, fonts, and documentation are also available in the Ubuntu repositories.

The `/usr` directory and its subdirectories contain the majority of the `xorg` software (along with a lot of other stuff; the location is not exclusive to X). Some important subdirectories are as follows:

- ▶ `/usr/bin`—This is the location of the X server and various X clients. (Note that not all X clients require active X sessions.)
- ▶ `/usr/include`—This is the path to the files necessary for developing X clients and graphics such as icons.
- ▶ `/usr/lib`—This directory contains software libraries required to support the X server and clients.
- ▶ `/usr/lib/x11`—This directory contains fonts, default client resources, system resources, documentation, and other files that are used during X sessions and for various X clients. You will also find a symbolic link to this directory, named `x11`, under the `/usr/lib` directory.

The main components required for an active local X session are installed on your system if you choose to use a graphical desktop. These components are the X server, miscellaneous fonts, a *terminal client* (that is, a program that provides access to a shell prompt),

and a client known as a *window manager*. Window managers administer onscreen displays, including overlapping and tiling windows, command buttons, title bars, and other onscreen decorations and features.

Elements of the `xorg.conf` File

Traditionally, the most important file for `xorg` has been the `xorg.conf` configuration file. This file used to contain configuration information that was vital for X to function correctly and was usually created during the installation of Ubuntu.

BULLETPROOF X

Ubuntu is designed to work no matter what might happen. So in the event of some cataclysmic event that destroys your main X system, you still have some graphical way of getting yourself back into a fully functional X-based system. *An additional upside is that much of the complexity of the information in this chapter is unnecessary for most users; in fact, the files are not even created or used by default and are used only if you create them.* The downside to this is that much of the configurability of the X server is now overwritten when an upgrade happens.

Modern versions of `xorg` do not create an `xorg.conf` file by default. Instead, various files ending in `*.conf` reside in the `/usr/share/X11/xorg.conf.d` directory and are automatically loaded by X at boot, prior to reading any `xorg.conf`. These files can each contain one or more sections in the same format used by `xorg.conf`. Users can create the file and continue making custom configurations in `/etc/xorg.conf` as has been traditionally done, but the file is not created by default. What is included in the previously mentioned individual files should not be changed, but you may override those settings by creating your own `/etc/xorg.conf` file.

NOTE

We refer to using an `xorg.conf` file from here on, but you should keep the preceding information in mind to prevent confusion.

Let's take a look at the potential contents of `xorg.conf` so that you can get an idea of what X is looking for. The components, or sections, of the `xorg.conf` file specify the X session or *server layout*, along with pathnames for files that are used by the server, any options related directly to the server, any optional support modules needed, information relating to the mouse and keyboard attached to the system, the graphics card installed, the monitor in use, and the resolution and color depth that Ubuntu uses. These are the essential components:

- ▶ **ServerLayout**—Defines the display, defines one or more screen layouts, and names input devices.
- ▶ **Files**—Defines the locations of colors, fonts, or port number of the font server.

- ▶ **Module**—Tells the X server what graphics display support code modules to load.
- ▶ **InputDevice**—Defines the input devices, such as the keyboard and mouse; multiple devices can be used.
- ▶ **Monitor**—Defines the capabilities of any attached display; multiple monitors can be used.
- ▶ **Device**—Defines one or more graphics cards and specifies what optional features (if any) to enable or disable.
- ▶ **Screen**—Defines one or more resolutions, color depths, perhaps a default color depth, and other settings.

The following sections provide short descriptions of these elements; the `xorg.conf` man page contains full documentation of all the options and other keywords you can use to customize your desktop settings.

The `ServerLayout` Section

As noted previously, the `ServerLayout` section of the `xorg.conf` file defines the display and screen layouts, and it names the input devices. A typical `ServerLayout` section from an automatically configured `xorg.conf` file might look like this:

```
Section "ServerLayout"
    Identifier      "single head configuration"
    Screen         0  "Screen0" 0 0
    InputDevice    "Mouse0" "CorePointer"
    InputDevice    "Keyboard0" "CoreKeyboard"
    InputDevice    "DevInputMice" "AlwaysCore"
EndSection
```

In this example, a single display is used (the numbers designate the positions of a screen), and two default input devices, `Mouse0` and `Keyboard0`, are used for the session.

The `Files` Section

The `Files` section of the `xorg.conf` file might look like this:

```
Section "Files"
    RgbPath        "/usr/lib/X11/rgb"
    FontPath       "unix/:7100"
EndSection
```

This section lists available session colors (by name, in the text file `rgb.txt`) and the port number to the X font server. The font server, `xfs`, is started at boot and does not require an active X session. If a font server is not used, the `FontPath` entry could instead list each font directory under the `/usr/lib/X11/fonts` directory, as in this example:

```
FontPath "/usr/lib/X11/fonts/100dpi"
FontPath "/usr/lib/X11/fonts/misc"
FontPath "/usr/lib/X11/fonts/75dpi"
```

```
FontPath "/usr/lib/X11/fonts/type1"
FontPath "/usr/lib/X11/fonts/Speedo"
...
```

These directories contain the default compressed fonts that are available for use during the X session. The font server is configured by using the file named `config` under the `/etc/x11/fs` directory. This file contains a listing, or catalog, of fonts for use by the font server. By adding an alternate-server entry in this file and restarting the font server, you can specify remote font servers for use during X sessions. This can help centralize font support and reduce local storage requirements (even though only 25MB is required for the almost 5,000 fonts installed with Ubuntu and X).

The Module Section

The `Module` section of the `xorg.conf` file specifies loadable modules or drivers to load for the X session. This section might look like this:

```
Section "Module"
    Load "dbe"
    Load "extmod"
    Load "fbdevhw"
    Load "glx"
    Load "record"
    Load "freetype"
    Load "type1"
    Load "dri"
EndSection
```

These modules can range from special video card support modules to font rasterizers. The modules are located in subdirectories under the `/usr/lib/modules` directory.

The InputDevice Section

The `InputDevice` section configures a specific device, such as a keyboard or mouse, as in this example:

```
Section "InputDevice"
    Identifier "Keyboard0"
    Driver "kbd"
    Option "XkbModel" "pc105"
    Option "XkbLayout" "us"
EndSection

Section "InputDevice"
    Identifier "Mouse0"
    Driver "mouse"
    Option "Protocol" "IMPS/2"
    Option "Device" "/dev/input/mice"
    Option "ZAxisMapping" "4 5"
    Option "Emulate3Buttons" "yes"
EndSection
```

You can configure multiple devices, and multiple `InputDevice` sections might exist. The preceding example specifies a basic keyboard and a two-button PS/2 mouse (actually, a Dell touchpad pointer). An `InputDevice` section that specifies use of a USB device could be used at the same time (to enable mouse use with PS/2 and USB pointers) and might look like this:

```
Section "InputDevice"
    Identifier "Mouse0"
    Driver "mouse"
    Option "Device" "/dev/input/mice"
    Option "Protocol" "IMPS/2"
    Option "Emulate3Buttons" "off"
    Option "ZAxisMapping" "4 5"
EndSection
```

The Monitor Section

The `Monitor` section configures the designated display device as declared in the `ServerLayout` section, as shown in this example:

```
Section "Monitor"
    Identifier "Monitor0"
    VendorName "Monitor Vendor"
    ModelName "Monitor Model"
    DisplaySize 300 220
    HorizSync 31.5-48.5
    VertRefresh 50-70
    Option "dpms"
EndSection
```

Note that the X server automatically determines the best video timings according to the horizontal and vertical sync and refresh values in this section. If required, old-style modeline entries (used by distributions and servers prior to XFree86 4.0) might still be used. If the monitor is automatically detected when you configure X, its definition and capabilities are inserted in your `xorg.conf` file from the `MonitorsDB` database. This database contains more than 600 monitors and is located in the `/usr/share/hwdata` directory.

The Device Section

The `Device` section provides details about the video graphics chipset used by the computer, as in this example:

```
Section "Device"
    Identifier "Intel Corporation Mobile 945GM/GMS,\943/940GML Express
Integrated Graphics Controller"
    Driver "intel"
    BusID "PCI:0:2:0"
EndSection
```

This example identifies an installed video card as using an integrated Intel 945 graphics chipset. The `Driver` entry tells the `Xorg` server to load the intel kernel module. Different chipsets have different options. For example, here's the entry for a NeoMagic video chipset:

```
Section "Device"
    Identifier    "NeoMagic (laptop/notebook) "
    Driver        "neomagic"
    VendorName    "NeoMagic (laptop/notebook) "
    BoardName     "NeoMagic (laptop/notebook) "
    Option        "externDisp"
    Option        "internDisp"
EndSection
```

In this example, the `Device` section specifies the driver for the graphics card (`neomagic_drv.o`) and enables two chipset options (`externDisp` and `internDisp`) to allow display on the laptop's LCD screen and an attached monitor.

The `Xorg` server supports hundreds of different video chipsets. If you configure X11 but subsequently change the installed video card, you need to edit the existing `Device` section or generate a new `xorg.conf` file, using one of the X configuration tools discussed in this chapter, to reflect the new card's capabilities. You can find details about options for some chipsets in a companion man page. You should look at these sources for hints about optimizations and troubleshooting.

The Screen Section

The `Screen` section ties together the information from the previous sections (using the `Screen0`, `Device`, and `Monitor Identifier` entries). It can also specify one or more color depths and resolutions for the session. Here's an example:

```
Section "Screen"
    Identifier    "Screen0"
    Device        "Videocard0"
    Monitor       "Monitor0"
    DefaultDepth  24
    SubSection    "Display"
        Viewport  0 0
        Depth     16
        Modes     "1024x768" "800x600" "640x480"
    EndSubSection
EndSection
```

In this example, a color depth of thousands of colors and a resolution of 1024×768 is the default, with optional resolutions of 800×600 and 64×480. Multiple `Display` subsection entries with different color depths and resolutions (with settings such as `Depth 24` for millions of colors) can be used if supported by the graphics card and monitor combination.

You can also use a `DefaultDepth` entry (which is 24, or thousands of colors, in the example), along with a specific color depth to standardize display depths in installations.

You can also specify a desktop resolution larger than that supported by the hardware in your monitor or notebook display. This setting is known as a *virtual* resolution in the `Display` subsection. This allows, for example, an 800×600 display to pan (that is, slide around inside) a virtual window of 1024×768.

Starting X

You can start X sessions in a variety of ways. The Ubuntu installer sets up the system to have Linux boot directly to an X session using a display manager called *GDM*, for GNOME Display Manager. This is an X client that provides a graphical login. After you log in, you use a local session (running on your computer) or, if the system is configured to do so, an X session running on a remote computer on the network.

If you log in via a display manager, you must enter a username and password. You can also start X sessions from the command line. The following sections describe these two methods.

NOTE

If you have used the Server install, your system boots to a text login. See Chapter 10, “Command-Line Beginner’s Class,” for more information about what to do here.

Using a Display Manager

An X display manager presents a graphical login that requires a username and password to be entered before access is granted to the X desktop. It also enables you to choose a different desktop for your X session.

Changing Window Managers

Ubuntu makes it fairly painless to switch to another window manager or desktop environment. *Desktop environment* refers to not only the window manager but also the suite of related applications, such as productivity or configuration tools.

You need to ensure that you have the relevant desktop environment installed on your system; the easiest way to do this is by installing the relevant `*-desktop` package. You can do this by installing the package `kubuntu-desktop`, for example (in the case of a KDE desktop); just search for “desktop” and look for Xubuntu or Kubuntu, and so on. After the download and installation is complete (you might want to grab a coffee while you wait because these packages include a ton of dependencies and take some time to download, install, and configure), you are all set to change environments.

Next, you need to log out of Ubuntu. When you return to the login page, select your name as usual and then select the session named for the desktop you want to use. Chapter 4 provides a brief introduction to some of the most common desktop environments available, including the default, GNOME.

References

- ▶ **www.x.org**—Curators of the X Window System
- ▶ **www.x.org/Downloads_mirror.html**—A list of mirror sites for downloading the source to the latest revision of X
- ▶ **www.xfree86.org**—Home of The XFree86 Project, Inc., which provided a graphical interface for Linux for nearly 10 years
- ▶ **<https://wiki.ubuntu.com/X>**—The place to get started when learning about X and Ubuntu
- ▶ **<https://wiki.ubuntu.com/X/Config>**—Great information about configuring X on Ubuntu

CHAPTER 4

Ubuntu Desktop Options

When you install Ubuntu, by default you use the GNOME graphical user interface (GUI). This is generally called a desktop. GNOME has specific strengths that appeal to many users. However, some of us have unique requirements or just like to tinker. This chapter discusses both GNOME and some of the alternatives to GNOME, with a brief mention of some of their strengths and weaknesses and information on how to install each. This is not a complete list of all the options available but rather a brief survey of some of the most popular options.

GNOME IS NOT THE ONLY OPTION

When you install standard desktop Ubuntu, you also install GNOME. Not only is this is not your only option but you are also not limited to having only one desktop environment installed at a time. Each of the options we discuss in this chapter may be installed alongside GNOME in standard Ubuntu, and you are allowed to choose which one to use each time you log in and which one to use by default. This makes testing a new option less risky because switching back to what you already know is simple.

Desktop Environment

Traditionally, a *GUI* for computers has used a desktop metaphor; specifically, the interface uses the idea of a physical office desk as a metaphor to make interacting with the computer simple to comprehend. As in a real-world office,

IN THIS CHAPTER

- ▶ Desktop Environment
- ▶ Using GNOME: A Primer
- ▶ KDE and Kubuntu
- ▶ Xfce and Xubuntu
- ▶ LXDE and Lubuntu
- ▶ MATE and Ubuntu MATE
- ▶ Ubuntu Budgie
- ▶ Ubuntu Kylin
- ▶ References

a computer desktop environment uses files to store documents that contain information that is necessary for office tasks to be done. Computer systems take the metaphor further by adding applications, programs that enable users to create, manipulate, and delete those documents as needed, much as might be done in the real world, but with greater efficiency and often with greater power.

A computer desktop includes a windowing system, another metaphoric way to deal with information that makes conceptualizing the complexity occurring behind the scenes in the digital realm easier. Files, folders (or directories), and applications open in a graphic display that may be moved around the user's desktop just as paper documents and file folders may be moved to different locations on a physical desktop for convenience and efficiency. This windowing system includes the graphical bits (called *widgets*) needed to draw the windows on the screen and all the program code needed to perform the actions desired behind the scenes while displaying those actions using the pretty metaphor of a physical desktop. For example, moving a document from one folder to another on a desktop involves little more than a mouse click followed by a drag and drop. Behind the scenes, the desktop environment is listening to the window manager's instructions to move a file from one directory to another and then telling the window manager to draw the event on the screen as it is accomplished.

The Ubuntu software repositories include all the popular desktop environments available for Linux and make installing and trying them out easy to do. It is also possible for an adventurous person to find other desktop environments that will work on Ubuntu and download and install them. We don't cover that process here.

It is also possible to use one desktop environment while substituting a different window manager into that environment instead of using the one that comes with the desktop environment by default. For example, the standard GNOME window manager is called Mutter. It is stable and works quite well. Other interfaces may use different window managers, such as Compiz. To make matters more interesting, you can replace either of those with other window managers such as Enlightenment, which does things in a unique manner, such as having menus pop up on the screen when you right-click rather than being anchored to a specific location.

This chapter focuses on complete desktop environments, using each one's default window manager. In addition, because this is an Ubuntu-focused book, this chapter concentrates on Ubuntu-refined versions of environments such as KDE and Xfce rather than their default versions, although the default versions are also available in the Ubuntu repositories in case you want to seek them out.

Using GNOME: A Primer

The default desktop in Ubuntu is GNOME 3. It is used not only by Ubuntu but also by several other Linux distributions, such as Debian, Fedora, Red Hat Enterprise Linux, and Oracle Linux.

The official GUI for GNOME 3 is called GNOME Shell. For this chapter, the only thing that has been changed from the default Ubuntu 18.04 installation is the background; the background used here is plain and simple to make the screen captures clear.

Upon logging in, the desktop is displayed (see Figure 4.1).

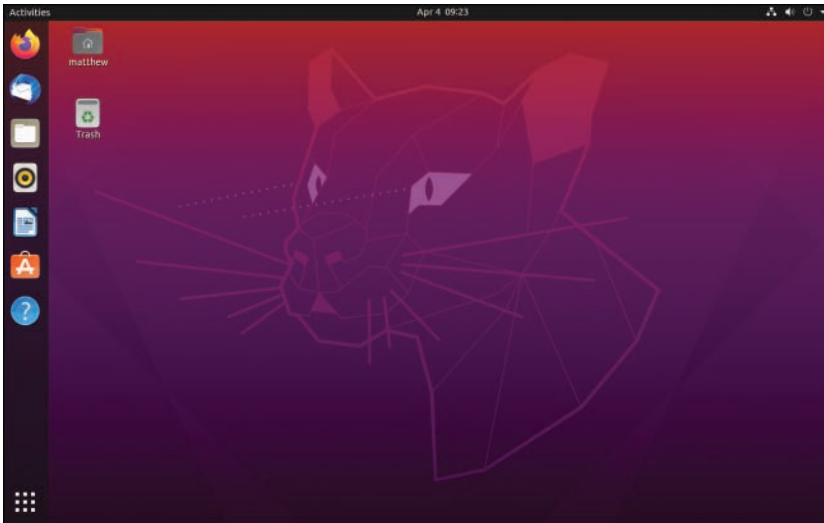


FIGURE 4.1 The GNOME Shell desktop.

From the desktop, you can do the following:

- ▶ Click Activities at the upper left to search the computer for installed programs.
- ▶ Select a program from the Dash (sometimes called the Favorites menu), which is the vertical tool bar on the left, starting just below the Activities link (see Figure 4.2). To add or remove an item from the Dash, right-click while hovering over an icon and select Add or Remove from the pop-up menu.
- ▶ Click the Show Applications icon at the bottom left (which looks like a square made of dots) to reveal an icon-rich array of installed programs (see Figure 4.3). Click any icon in this list to open the program.
- ▶ Click the power icon at the upper right to reveal a list of networking, settings, screen lock, and shutdown options (see Figure 4.4).

At the top center of the desktop you see a day and time that you can click to reveal a calendar. Any notifications you have from Ubuntu are also displayed and available here.

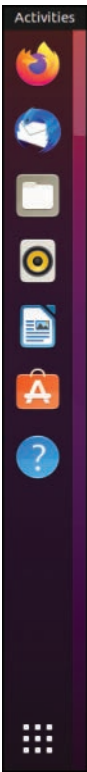


FIGURE 4.2 The GNOME Dash, with the Activities link at the top.



FIGURE 4.3 Click the Show Applications icon to reveal an array of installed programs.

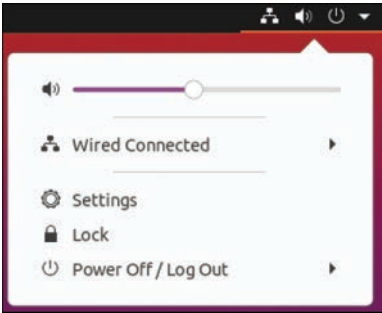


FIGURE 4.4 Adjust volume, account settings, and much more from the power menu.

KDE and Kubuntu

The KDE project began back in 1996, with the goal of creating a quality desktop environment for Linux that was free, worked well, and was well integrated, meaning that programs and interfaces would have a consistent look and feel, with all the parts fitting together seamlessly rather than looking like a jumbled compilation of a bunch of assorted bits.

The KDE project has always been focused on end users rather than creating a simple GUI for the system administrator. This is an important point because since the beginning, the intent has been to make any computer user comfortable and able to do what he or she wants to do without necessarily requiring a full grasp of what is happening behind the scenes. This focus continues today and is shared by other desktop environments, notably GNOME, which was started in 1997 by people once involved in the KDE project, who left after a dispute over software licensing.

The cause of that dispute no longer exists as the licenses today are equivalently free, but the projects diverged a bit in their focus: GNOME offers a desktop of zenlike simplicity with simple and elegant defaults and use, whereas KDE presents more flash and configuration options. Honestly, they are both great desktop environments that are well integrated with a high degree of professionalism and quality and are easily the top two in the Linux world. If you have never used either, try them both and see which you prefer.

Kubuntu is a project that started in the Ubuntu community very early, on with the simple goal of enabling users to install and use KDE in Ubuntu. To make this even easier for people who already know they prefer KDE over GNOME, you can download and install disk for Kubuntu, which is Ubuntu minus GNOME plus KDE plus a few Kubuntu-specific enhancements. You may also install Kubuntu in standard Ubuntu and alongside GNOME by installing the `kubuntu-desktop` package from the Ubuntu software repositories.

Kubuntu (see Figure 4.5) uses a different set of default programs for most tasks: web browsing, email, and so on. Most were written specifically for the KDE desktop environment, making KDE one of the most closely integrated and consistent Linux desktops available.



FIGURE 4.5 The Xubuntu desktop.

Xfce and Xubuntu

Xfce is a lighter desktop environment that requires less memory and processing power than either GNOME or KDE and is therefore often suggested for use on older machines. It does not deserve to be relegated to that role because it also works great on current hardware and is often a quality choice for a different reason. Xfce has been developed using the traditional UNIX philosophy of software: Do one thing, do it well, and play well with others.

The parts of the Xfce desktop environment are modular; you can add or remove bits at will and substitute other programs that perform the same function. Everything created by or included in Xfce is expected to work according to simple standards that allow other programs to interact with it easily.

On one hand, this sometimes means that people look at Xfce and think it isn't as seamlessly integrated and smooth as the two main desktop environments. On the other hand, it means that if you prefer the GNOME file manager (Nautilus) over the one included with Xfce (Thunar), you can just install Nautilus and use it side-by-side with Thunar or remove Thunar completely. This is a huge part of what makes Xfce so lightweight that it has very few dependency requirements and is highly flexible.

Originally, Xubuntu (see Figure 4.6) was designed to create a lighter-weight version of Ubuntu that would run well on older hardware because of the lighter code dependencies of Xfce. Over time, some people discovered that they liked the desktop environment for

other reasons, and the older hardware use case became less of a focus. It was the modularity of Xfce combined with a smoothness of operation that won people over, and the distribution began to take some of the favored bits from Ubuntu's customized version of GNOME and added them to Xfce to replace some of its defaults. What we have today is a nice amalgamation of Ubuntu GNOME bits, Xfce bits, and a few other things not included by default in either.



FIGURE 4.6 The Xubuntu desktop.

Xubuntu still uses less memory and fewer CPU cycles than a standard Ubuntu or Kubuntu install; however, thinking of it only in those terms doesn't do it justice. To install Xubuntu with the Xfce desktop environment, install the `xubuntu-desktop` package.

LXDE and Lubuntu

Lubuntu is based on LXDE, an extremely fast desktop environment that uses less memory and fewer CPU cycles than any of the others discussed. It is developed specifically with lower-powered computers such as netbooks in mind, but that isn't the sole use case. For example, Knoppix, which is a Linux distribution that runs from a live, bootable CD or DVD, now uses LXDE. Knoppix is a longtime favorite of sysadmins for emergency repairs of unbootable systems and for its portability. It recently switched from KDE to LXDE to benefit from this lightness because running an operating system from a CD or DVD is generally much slower than when it is installed on a hard drive.

As the focus in Xubuntu turned from speed and lightness to enjoying the flexibility of Xfce, a gap was created. Users and developers interested in less-expensive hardware, such as mobile Internet devices and ARM or MIPS processor-based computers, wanted to find a way to run a distribution of Linux that shared the community of Ubuntu, that had a beautiful and useful desktop, and that did not get bogged down on slower machines. LXDE is quite new, and its development philosophy fits quite well with the hopes and dreams of these users, so it seems a perfect fit.

The Lubuntu (see Figure 4.7) distribution is still very new. The developers are working within the Ubuntu community and making consistent progress, and they are also appealing for interested people to join the development team and help out. Install `lubuntu-desktop` from the Ubuntu repositories to check it out.

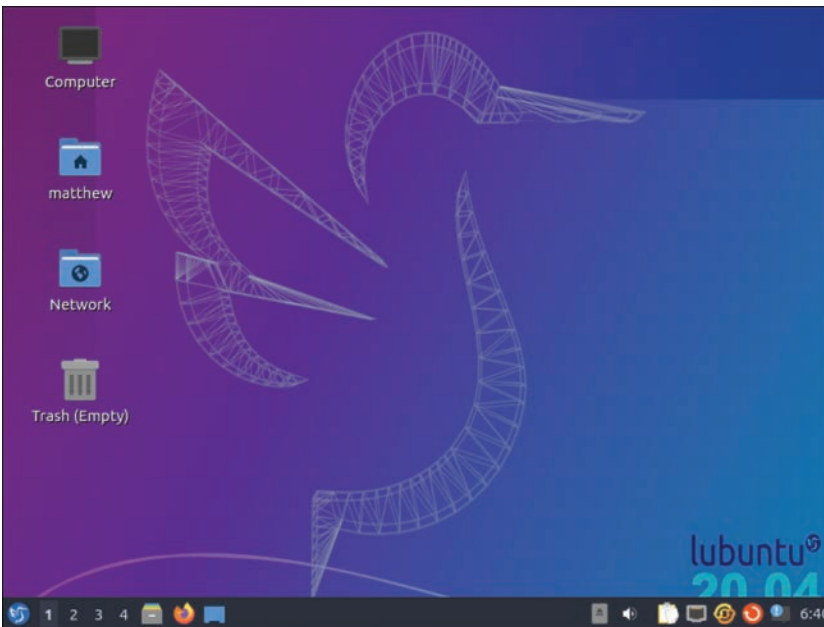


FIGURE 4.7 The Lubuntu desktop is quite attractive.

MATE and Ubuntu MATE

GNOME 2 was well loved, but it is no longer supported and therefore not recommended. However, its code was forked into MATE (pronounced like the tea-like drink from South America, “mah-tay”). Basically, MATE is a continuation of GNOME 2, the older version of GNOME before GNOME 3 was released. MATE has the features, look, and feel of the older version of GNOME and the same code foundation but with continued development (see Figure 4.8).

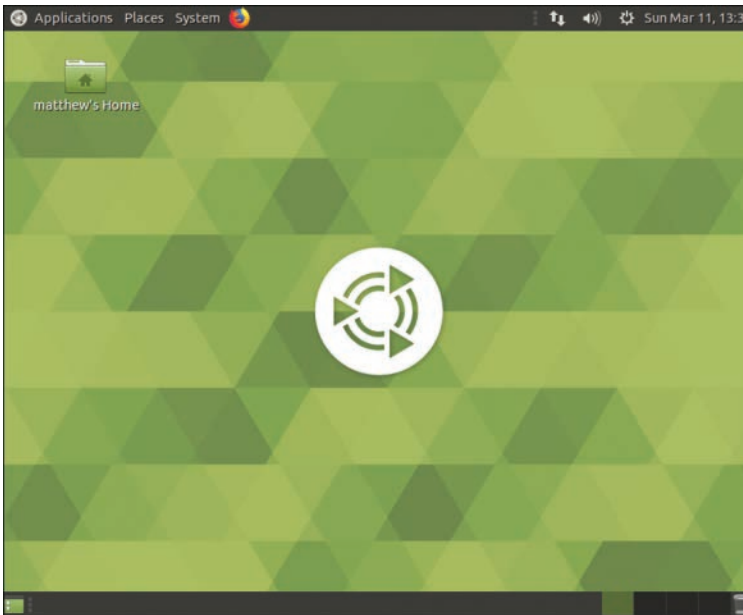


FIGURE 4.8 The MATE desktop.

Ubuntu Budgie

Ubuntu Budgie uses the Budgie desktop (see Figure 4.9) from <https://budgie-desktop.org>. It is a new spin on desktops in general and is designed for simplicity and elegance. It is beautiful and interesting, and while it started back in 2013, it has just begun to reach maturity and genuine usefulness over the past few years. It is well worth a look!



FIGURE 4.9 The Budgie desktop.

Ubuntu Kylin

Ubuntu Kylin is Ubuntu localized for China. It starts out the same as standard Ubuntu and is modified with Chinese language, calendar, and cultural customizations for the world's largest market. As it is in a different language than this book, we are only giving it a cursory mention. However, if you are a Chinese speaker, you will probably find this interesting and worth a look.

References

- ▶ **www.gnome.org**—The official site of GNOME
- ▶ **www.compiz.org**—The official site of Compiz
- ▶ **www.enlightenment.org**—The official site of Enlightenment
- ▶ **www.kde.org**—The official site for the KDE desktop
- ▶ **www.kubuntu.org**—The official Kubuntu site
- ▶ **www.xfce.org**—The official site for the Xfce desktop
- ▶ **www.xubuntu.org**—The official Xubuntu site
- ▶ **www.lxde.org**—The official site for the LXDE desktop
- ▶ **http://lubuntu.net**—The official Lubuntu site
- ▶ **www.gnome.org**—The official site for the GNOME desktop
- ▶ **http://ubuntubudgie.org**—The official Ubuntu Budgie site
- ▶ **https://ubuntu-mate.org**—The official Ubuntu MATE site
- ▶ **https://wiki.ubuntu.com/UbuntuKylin**—The English language resource for Ubuntu Kylin
- ▶ **http://www.ubuntu.com/project/about-ubuntu/derivatives**—The official Ubuntu list of recognized derivatives
- ▶ **www.knoppix.net**—The site for the live CD/DVD Linux distribution, Knoppix
- ▶ **http://en.wikipedia.org/wiki/Desktop_environment**—A detailed definition of *desktop environment*, how they work, why they exist, and so on
- ▶ **http://en.wikipedia.org/wiki/Comparison_of_X_Window_System_desktop_environments**—An amazing list of available desktop environments, complete with a comparison of features

CHAPTER 5

On the Internet

The Internet is used everywhere. From cell phones to offices, from game consoles to tablets, we are surrounded by multiple access routes to online information and communication. Ubuntu is no outsider when it comes to accessing information through the Internet; it comes equipped with all the tools you need to connect to other people around the globe.

A BRIEF INTRODUCTION TO THE INTERNET

The Internet itself was first brought to life by the U.S. Department of Defense in 1969. It was called ARPANET, after the Department of Defense's Advanced Research Projects Agency. Designed to build a network that would withstand major catastrophe (this was the peak of the Cold War), the agency soon grew to encompass more and more networks to build the Internet, essentially a massive network comprised of all these other networks. Then, in 1991, Tim Berners-Lee of CERN developed the idea of the World Wide Web, including *Hypertext Transfer Protocol (HTTP)* and *Hypertext Markup Language (HTML)*. What most people think of today as "the Internet" is really just part of it, the World Wide Web.

The Internet used to mean using many applications and protocols to interact with others, acquire information, and do things. Programs such as Archie and Gopher were used to retrieve information, elm and pine were used for interacting with email, and Usenet was the place for interacting with others via the first worldwide messaging board. All of these have been long surpassed by one protocol, HTTP, and one idea, the World Wide Web.

IN THIS CHAPTER

- ▶ Getting Started with Firefox
- ▶ Checking Out Google Chrome and Chromium
- ▶ References

With that in mind, we don't cover the older technologies that most people will never use that were covered in previous editions of this book. Most interaction today is via the Web. Entire operating systems have been superseded in some cases by just a minimal OS layer and a platform powered by web access, as in inexpensive laptops running Chrome OS. So instead of covering antiquated programs, we present a short chapter on some web browser options.

Getting Started with Firefox

One of the most popular web browsers, and, in fact, the default web browser in Ubuntu, is Mozilla Firefox (see Figure 5.1). Built on a solid code base that is derived from the Mozilla suite, Firefox offers an open source and secure option for surfing the Internet, regardless of operating system.

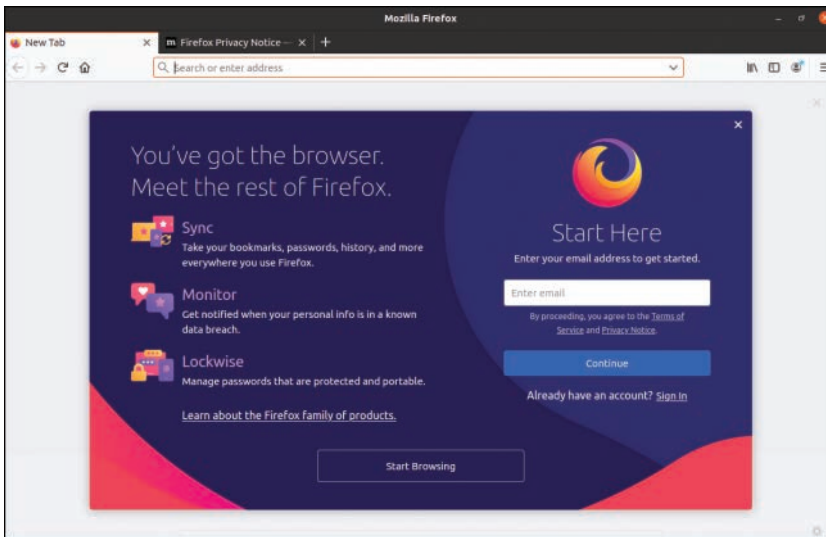


FIGURE 5.1 Mozilla Firefox enables you to add numerous extensions to further enhance your web experience.

In Ubuntu, you can find Firefox from the sidebar or by searching under Activities for "firefox."

Beyond the basic program are a plethora of plug-ins and extensions that can increase the capabilities of Firefox beyond simple web browsing. Plug-ins such as Java are available for installation instantly, offered to you when first needed, as are multimedia codecs for viewing video content. Extensions provide useful additions to the browsing experience. For example, ForecastFox is an extension that provides your local weather conditions, and Bandwidth Meter and Diagnostics is a tool that calculates your current bandwidth.

There are many more extensions and plug-ins that you can use to enhance your browsing experience, and you can find them by pressing Shift+Ctrl+A while in Firefox.

You can find and obtain plug-ins and extensions easily because Mozilla developers have created a site dedicated to helping you get more from Firefox. You don't have to search to find the site as there is a link in the Firefox menu at Tools, Add-ons, Get Add-ons that takes you directly to it. A particular favorite is the Adblock Plus plug-in, which enables you to nuke all those annoying banners and animations that take up so much bandwidth while you are browsing.

Checking Out Google Chrome and Chromium

The most popular browser today is Google Chrome (see Figure 5.2), which also runs on Linux. This browser is fast, secure, expandable with extensions, and the hottest new toy on geek desktops. It is based on WebKit and other technologies, and it downloads and renders quickly while being compliant with standards. You can learn more about the easy installation of Google Chrome at www.google.com/chrome.

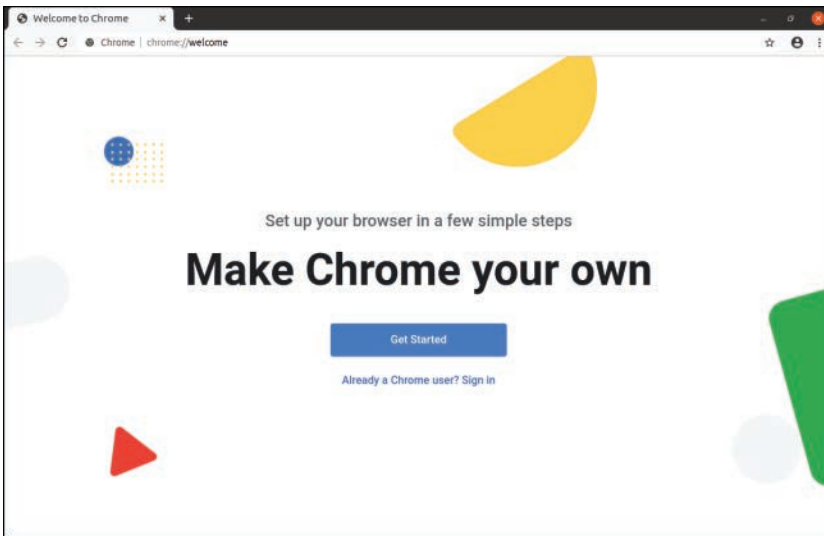


FIGURE 5.2 Google Chrome running in Ubuntu.

Chrome is not completely open source, but the main foundational parts are open source and are made available as Google Chromium, which has most of the functionality of Chrome but is missing the Google branding and automatic updates; it is used as the test and development foundation for Chrome releases. (As a point of interest, chromium is the metal from which chrome is made—hence the names.)

You can learn more about Chromium at www.chromium.org. You can install `chromium-browser` from the Ubuntu repositories.

References

- ▶ **www.mozilla.com**—The home page for Firefox
- ▶ **www.google.com/chrome**—The home page for Chrome
- ▶ **www.chromium.org/Home**—The home page for Chromium

CHAPTER 6

Productivity Applications

Many businesses have already found a way to benefit from free and open source software, such as office productivity suites like LibreOffice, instead of paying license fees for business use of either traditional locally installed options or web apps. However, more applications beyond these are available in Ubuntu. In this chapter, we explore some of them.

NOTE

It's important to understand that even though free and open source software does very well most of the time, especially with less-complex documents, it is not 100 percent compatible with Microsoft Office. Why is this? Microsoft is notoriously secretive about its proprietary file formats, and the only way that free and open source alternatives could ensure compatibility would be to reverse-engineer each file format, an exercise akin to taking apart a telephone to see how it works. This reverse-engineering is difficult to do in a legal way and is rarely perfect. However, many open source alternatives manage to maintain a very high standard of importing and exporting, so you should not experience too many problems except with documents of great complexity. For example, this book was written using LibreOffice, whereas the post-production at the publisher uses Microsoft tools.

The biggest compatibility issue between Microsoft Office and suites like LibreOffice is that Microsoft's *Visual Basic for Applications (VBA)* and scripts produced using it do not always transfer. If you use VBA scripts, you should test as most are compatible, but occasionally you may need to find another way to perform the functions for which they were written.

IN THIS CHAPTER

- ▶ Introducing LibreOffice
- ▶ Other Useful Productivity Software
- ▶ Productivity Applications Written for Microsoft Windows
- ▶ References

A *productivity suite* is defined as two or more applications bundled together and used for creating documents, presentations, spreadsheets, and databases. Other applications that might be included in such a bundle are email clients, calculators/formula editors, and illustration or drawing software. Commonly, they are all tied together with a common look and feel, which makes sticking to one particular suite much easier. Because Ubuntu uses LibreOffice as its standard office suite, we introduce you to that first. We also take a brief look at some of the other Linux-based productivity applications.

PRODUCTIVITY FOR THE TYPICAL USER

For the majority of users of productivity suites, LibreOffice should fulfill most, if not all, requirements. However, the first hurdle is not whether it can do what you require of it but rather whether it can successfully import and export to proprietary Microsoft formats at a standard that is acceptable to your needs. Most of the time, LibreOffice should import and export with minimal hassle, perhaps getting a bit stuck with some of the more esoteric Microsoft Office formatting. Given that most users do not go much beyond tabs, columns, and tables, this level of compatibility should suffice.

However, you are strongly advised to round up a selection of documents and spreadsheets that seem the most likely to be difficult for the import/export filter and test them thoroughly (of course, keeping a backup of the originals). A system administrator who has deployed a new productivity suite does not want to suddenly get users complaining that they cannot read their files. This would quickly destroy any benefits gained from the other useful functions within LibreOffice and could even cause a return to proprietary formats and expensive office suites.

On the positive side, LibreOffice supports a huge array of file formats and can export to nearly 70 types of documents. Such a variety of file formats means that you should be able to successfully use LibreOffice in nearly any environment, including formats no longer used by currently produced and maintained software, so it may be able to open some old files and documents you had once given up for lost.

Introducing LibreOffice

LibreOffice contains a number of productivity applications for use in creating text documents, preparing spreadsheets, organizing presentations, managing projects, and so on. The following components of the LibreOffice package are included with Ubuntu:

- ▶ **Writer**—This word processing program enables you to compose, format, and organize text documents. If you are accustomed to using Microsoft Word, the functionality of LibreOffice Writer will be familiar to you.
- ▶ **Calc**—This spreadsheet program enables you to manipulate numbers in a spreadsheet format. Support for all but the most esoteric Microsoft Excel functions means that trading spreadsheets with Excel users should be successful. Calc offers some limited compatibility with Excel macros, but those macros generally have to be rewritten.

- ▶ **Impress**—This presentation program, which is similar to Microsoft PowerPoint, enables you to create slideshow presentations that include graphs, diagrams, and other graphics. Impress also works well with most PowerPoint files.

NOTE

The following five applications are not included by default with Ubuntu but are quite useful. All but Dia are a part of the LibreOffice project and add features to the suite that are not used as often as those Ubuntu installs by default. You must install them from the Ubuntu repositories if you want or require their functionality.

- ▶ **Math**—This math formula editor enables you to write mathematical formulas with a number of math fonts and symbols for inclusion in a word processing document. Such symbols are highly specialized and not easily included in the basic functionality of a word processor. This is of interest primarily to math and science writers, but Math can be useful to anyone who needs to include a complex formula in text.
- ▶ **Base**—This is a fully functional database application.
- ▶ **Draw**—This graphics application allows you to create images for inclusion in the documents produced with LibreOffice. It saves files only in LibreOffice format, but it can import most common image formats.
- ▶ **Dia**—This technical drawing editor enables you to create measured drawings, such as those used by architects and engineers. Its functionality is similar to that of Microsoft Visio.
- ▶ **Planner**—You can use this project management application for project planning, scheduling, and tracking; this application is similar to, but not compatible with, Microsoft Project. After you install it, you will find it under the name Project Management.

A BRIEF HISTORY OF LIBREOFFICE

LibreOffice started as a fork of the OpenOffice.org office suite. The OpenOffice.org office suite is based on a commercial suite called StarOffice. Originally developed by a German company, StarOffice was purchased by Sun Microsystems in the United States. One of the biggest complaints about the old StarOffice was that all the component applications were integrated under a StarOffice “desktop” that looked very much like a Microsoft Windows desktop, including a Start button and menus. This meant that to edit a simple document, unneeded applications had to be loaded, making the office suite slow to load, slow to run, and quite demanding on system resources.

After the purchase of StarOffice, Sun Microsystems released a large part of the StarOffice code under the GNU General Public License, and development began on what has become OpenOffice.org, which was freely available under the GPL. Sun also continued development on StarOffice. The significant differences between the free and commercial versions of the software were that StarOffice provided more fonts and even more import/export file filters than OpenOffice.org (these filters were not provided in the GPL version

because of licensing restrictions), and StarOffice provided its own relational database, Software AG's Adabas D database. Sun was bought by Oracle. Oracle suffered from a major disagreement with the developer community surrounding OpenOffice.org, and the developers left to form The Document Foundation, hoping that Oracle would eventually join. Because the code for OpenOffice.org was licensed using a free software license, The Document Foundation created a fork, or a new version of the same software, using what they intended as a temporary name, LibreOffice. The hope was merely to change how the project was governed, from being led by one company to being led by a community with many companies and individuals participating. Oracle chose not to join The Document Foundation and instead relicensed the OpenOffice.org code for all future versions, which it may do as the owner of that code, and gave the code to the Apache Software Foundation, which is licensing it under the less-restrictive Apache license that allows open source code to be used in proprietary products. To make things more interesting, IBM is using this Apache-licensed version of OpenOffice.org as the foundation for its own free (in terms of cost) office suite based on it, called Lotus Symphony, which also has some proprietary additions.

As the saga continues, the ultimate winner may be the end user as there are now effectively three competing office suites. For now, LibreOffice has the most developers, the strongest community, and the most mature software with the most rapid addition of new or improved features.

Other Useful Productivity Software

The office suites already discussed in this chapter are ideal for typical office-focused file interactions: creating basic documents, spreadsheets, and so on. However, some users have more complex or precise needs. This section covers some of the options available to help you be productive in those instances.

Working with PDFs

Reading a PDF in Ubuntu is simple. The functionality is available by default, thanks to an installed program called Evince. If you open a PDF, it opens in Evince, where you can read the document. Sometimes filling out forms is less straightforward as the form might have been created using functionality available only with Adobe. You can install Adobe Reader from the Ubuntu Software Center from the Canonical Partners section. Adobe Reader should work with any PDF form created using Adobe software, whether it was created on Windows, macOS, or Linux.

On occasion, you may have a PDF file that you want to edit. That is a little more complex than reading PDFs, but it's not as difficult as it used to be. There is a program created just for editing PDF files: PDF Edit (`pdfedit`).

Install `pdfedit` from the Ubuntu Software Center. On the surface, this program seems simple, but it has great power that is not immediately obvious. Advanced users can learn to use `pdfedit` in scripts to make sweeping changes quickly. Of course, as with most other powerful tools, it comes with the cost of complexity and a learning curve.

Writing Scripts

While you can write scripts using any standard word processing program, Celtx is a complex and powerful tool for serious writers and production managers. This is useful because scripts require keeping track of more than just words.

The application comes with options to help you organize your scenes; annotate those scenes with notes; create index cards; maintain your list of characters, locations, and props; and far more.

At first glance, Celtx seems overwhelming, but due to the inclusion of some sample projects, learning how to use it is not as difficult as it sounds. Install `celtx` from the Ubuntu Software Center.

Working with XML and DocBook

Like its ancestor SGML and cousin HTML, XML is a markup language. It is designed for use in a plain-text document. Tags surround specific sections of the text to denote how those sections are to be displayed. Listing 6.1 contains a short example.

LISTING 6.1 Sample XML Excerpt

```
<?xml version="1.0"?>
<xml-stylesheet type="text/css" href="book.css"?>
<book>
<title>Ubuntu Unleashed 2013</title>
<edition>8</edition>
<chapter>
  <number>1</number>
  <title>Installing Ubuntu</title>
  <text>
    <paragraph><dropcap>N</dropcap>ot that long ago, the mere mention...
    </paragraph>
    ...
  </text>
</chapter>
...
</book>
```

This could easily be written using a simple text editor like the one installed by default, `gedit` (called Text Editor in the Dash and the listings in the Ubuntu Software Center). However, doing it that way would be tedious for most people. A better option is to use an editor expressly designed and intended for dealing with XML files. Because DocBook is an open source standard form of XML that has been designed explicitly for use with documentation, many editors that can work with one will work with both. If you only need to do something quick, one of these should be suitable. Start with `gedit`, which is installed by default. If it is not suitable, look in the Ubuntu Software Center for other options, like the ones discussed next.

If you intend to write a lot of complicated documentation in only the DocBook format, the most common recommendation for Ubuntu is a program called Publican. Publican is not just an editor; it is also a publication system for DocBook. It tests your XML to ensure that it is in a valid DocBook form so that your output conforms to publication standards. It automates output into multiple formats such as HTML and PDF, and it allows complete control for custom skinning and formatting. You can install Publican from the Ubuntu Software Center.

A more powerful option is XML Copy Editor. It is designed for editing most markup languages, including XML, DocBook, DITA, and more. It also features schema validation, syntax highlighting, tag completion, and spell checking. This is the most useful option for a professional documentation specialist. You can install XML Copy Editor from the Ubuntu software repositories, where you can find a version for use on Windows. See <http://xml-copy-editor.sourceforge.net>.

Working with LaTeX

LaTeX was created for and is widely used in academia. It is a WYGIWYW (“what you get is what you want”) document markup language created for the TeX typesetting system. Multiple editors are available for use with LaTeX, and they are likely to be found for just about every operating system in existence.

NOTE

WYSIWYG is an acronym for “what you see is what you get” that has often been used to describe word processors and document creation systems that use a graphical interface. Unfortunately, anyone who has created documents with these programs, including the ones mentioned earlier in this chapter, such as LibreOffice, knows that what you see on the screen is not always what appears in the printed version on paper. There are no promises about how things will or will not look on the screen while using a LaTeX editor for your TeX document, but the format promises that the ultimate output will be exactly what you ask for.

A couple of the more popular LaTeX editors available from the Ubuntu Software Center are discussed in this section. You can also create and edit using any text editor, including `gedit`.

Texmaker not only has a version in the Ubuntu Software Center but also offers versions for Windows and macOS from www.xmlmath.net/texmaker/. It is free, easy to use, and mature. The program has been around for a while, and it is stable, has many useful features, and is rather popular in the TeX world.

LyX follows suit with both a version in the Ubuntu Software Center and versions available for Windows and macOS from its website, at www.lyx.org. The main appeal for LyX users is its graphical interface, which makes it an interesting bridge from WYSIWYG to LaTeX. It also has many plug-ins available to expand functionality.

Kile was written and designed for use with KDE. As such, it blends in well with Kubuntu but also runs well on a standard Ubuntu installation. It also has a Windows version available. See <https://kile.sourceforge.net> for details.

Creating Mind Maps

Sometimes the best way to help you gather your thoughts and ideas is to create a mind map. Heimer helps you do exactly that quickly and with an interface designed for simplicity and speed. There are not a ton of options. Instead, the focus is on providing blocks that you can fill in with text, color code if you want to, then drag and drop the blocks, associate them with other blocks, and so on to create the map you need. You can even export the maps to be used in other applications. Install `heimer` from the Ubuntu Software Center.

Productivity Applications Written for Microsoft Windows

Microsoft Windows is fundamentally different from Linux, yet you can install and run some Microsoft Windows applications in Linux by using an application named Wine. Wine enables you to use Microsoft Windows and DOS programs on UNIX-based systems. Wine includes a program loader that you can use to execute a Windows binary, along with a DLL library that implements Windows command calls, translating them to the equivalent UNIX and X11 command calls. Because of frequent updates to the Wine code base, Wine is not included with Ubuntu. Download a current version of Wine from www.winehq.org. To see whether your favorite application is supported by Wine, you can look at the Wine application database at <https://appdb.winehq.org>.

Other solutions, primarily CrossOver Office from CodeWeavers, enable use of Microsoft productivity applications. If you are after a closer-to-painless way of running not only Microsoft Office but also Apple iTunes and other software, you should investigate CodeWeavers. CrossOver Office is one of the simplest programs you can use to get Windows-based programs to work. Check out www.codeweavers.com to download a trial version of the latest software.

References

- ▶ **www.libreoffice.org**—The home page for the LibreOffice suite
- ▶ **www.documentfoundation.org**—The home page for The Document Foundation
- ▶ **www.openoffice.org**—The home page for the OpenOffice.org office suite
- ▶ **www.pdfedit.cz/en/index.html**—The home page for PDF Edit
- ▶ **www.codeweavers.com**—The home page for CrossOver Office from CodeWeavers that enables you to run some Windows programs under Linux

This page intentionally left blank

CHAPTER 7

Multimedia Applications

The twenty-first century has become the century of the digital lifestyle, with millions of computer users around the world embracing new technologies. It started with the rise of digital cameras, MP3 players, and other assorted multimedia gadgets. Fifteen years ago you might have had a small collection of WAV files scattered about your Windows installation, and ten years ago you were more likely to have hundreds, if not thousands, of MP3 files scattered across various computers. Today, most people enjoy music and video via their web browsers or their mobile phones, consuming from sites like YouTube and Spotify.

If you are a media creator or collector, however, you may have a large collection of audio files, video clips, animations, and other graphics. In this case, organizing and maintaining these vast libraries is not a problem.

This chapter provides an overview of some of the basic multimedia tools included with or available for Ubuntu.

Sound and Music

Linux once had a reputation for lacking good support for sound and multimedia applications in general. However, this really isn't true anymore and hasn't been for years. (It might make you smile to know that while Microsoft no longer supports the Microsoft Sound Card, Linux users still enjoy support for it—no doubt just to annoy the folks in Redmond.) UNIX, however, has always had good multimedia support, as David Taylor, UNIX author and guru, points out:

The original graphics work for computers was done by Evans and Sutherland on UNIX systems. The innovations at MIT's Media Lab were done on UNIX workstations. In 1985, we at HP Labs were creating sophisticated multimedia

IN THIS CHAPTER

- ▶ Sound and Music
- ▶ Graphics Manipulation
- ▶ Using Digital Cameras with Ubuntu
- ▶ Burning CDs and DVDs in Ubuntu
- ▶ Viewing Video
- ▶ Recording and Editing Audio
- ▶ Editing Video
- ▶ References

immersive work environments on UNIX workstations, so maybe UNIX is more multimedia than suggested. Limitations in Linux support doesn't mean UNIX had the same limitations. I think it was more a matter of logistics, with hundreds of sound cards and thousands of different possible PC configurations.

That last sentence sums it up quite well: UNIX had a limited range of hardware to support, whereas Linux has hundreds of sound cards. Sound card device driver support has been long lacking from manufacturers, and there is still no single standard for the sound subsystem in Linux.

In this section, you learn about sound cards, sound file formats, and the sound applications provided with Ubuntu.

Sound Cards

Ubuntu supports a wide variety of sound hardware and software. Two models of sound card drivers compete for prominence in today's market:

- ▶ ALSA, the *Advanced Linux Sound Architecture*, which is entirely open source
- ▶ OSS, the *Open Sound System*, which offers free and commercial drivers

Ubuntu uses ALSA because ALSA is the sound architecture for the Linux kernel, starting with the 2.6 series, all the way to the current 5.x series. OSS might still be found here and there, but it is no longer in widespread use and should be considered deprecated.

ALSA supports a long list of sound cards. You can review the list at www.alsa-project.org/main/index.php/Main_Page if you are interested, but Ubuntu detects most sound cards during the original installation and should detect any new additions to the system during boot. To configure the sound card at any other time, use the sound preferences graphical tool by searching the Dash for "sound."

In addition, Ubuntu uses an additional layer of software called *PulseAudio*. PulseAudio, which is a sound server, acts as a mediator between the various multimedia programs that have sound output and the ALSA kernel drivers. Over the years, many different sound servers have been used in Linux, each with different strengths, usability issues, and levels of documentation. These various sound servers have often been forced to run side by side on the same computer, causing all sorts of confusion and issues. PulseAudio aims to replace all of them and work as a single handler to accept output from applications that use the APIs for any of the major sound servers already in use, such as ESD, OSS, GStreamer, and aRts, and route the various output streams together through one handler. This provides several advantages, including the ability to control the output volume of various programs individually.

PulseAudio has matured over the past several releases and is better and more powerful than ever before. Although there were stability issues and complaints early on, they don't seem to be problematic anymore except in unusual hardware combinations and special cases, and more and more features have been implemented. For more information about PulseAudio, see www.pulseaudio.org.

Sound Formats

A number of formats exist for storing sound recordings. Some of these formats are associated with specific technologies, and others are used strictly for proprietary reasons. Ubuntu supports all the most popular sound formats, including the following:

- ▶ **RAW (.raw)**—RAW is more properly known as *headerless format*, and audio files using this format contain an amorphous variety of specific settings and encodings. All other sound files contain a short section of code at the beginning—a header—that identifies the format type.
- ▶ **MP3 and MP4 (.mp3, .mp4)**—These popular commercially licensed formats for the digital encoding are used by many Linux and Windows applications.
- ▶ **WAV (.wav)**—This popular uncompressed Windows audiovisual sound format is often used as an intermediate file format when encoding audio.
- ▶ **Ogg-Vorbis (.ogg)**—Ogg is the free software community's preferred audio encoding format. You enjoy better compression and audio playback and freedom from proprietary licensing concerns when you use this open source encoding format for your audio files.
- ▶ **FLAC (.flac)**—This lossless format is popular with audiophiles. The name stands for *Free Lossless Audio Format*, and it is a compressed format, like MP3, but does not suffer from any loss of quality.

Ubuntu also includes software in the repositories (such as the `sox` command, used to convert between sound formats) so that you can more easily listen to audio files provided in a wide variety of formats, such as AU (from NeXT and Sun), AIFF (from Apple and SGI), IFF (originally from Commodore's Amiga), RA (from Real Audio), and VOC (from Creative Labs).

TIP

For an introduction to audio formats, check out the list of audio file formats at www.fileinfo.com/filetypes/audio, which provides links to detailed information for the various formats.

Ubuntu also offers several utilities for converting sound files from one format to another. Conversion utilities come in handy when you want to use a sound in a format not accepted by your current application of choice. The easiest one to use is also the easiest to install. Install the `soundconverter` package and then search for Sound Converter. It has a clear graphical interface and easy-to-understand configuration options.

Listening to Music

If you're anything like us, you might be a huge music fan. Beyond using your web browser with your favorite sites, which is simple and common, you can also play music without an Internet connection.

Rhythmbox

Rhythmbox is a useful application that plays CDs and music files. It can also rip music from CDs into your favorite format.

Banshee

Banshee is another music application that can handle ripping and playing back music, download cover art, sync with portable players, and even play video.

GETTING MUSIC INTO UBUNTU WITH SOUND JUICER

A handy utility that is included with Ubuntu is Sound Juicer, found under Applications, Sound & Video as the Audio CD Extractor. Sound Juicer automatically detects when you install a CD and attempts to retrieve the track details from the Internet. From there, it rips the CD tracks into .ogg files for storage on your file system.

Graphics Manipulation

Over a short period of time, digital cameras, camera apps on mobile phones, and digital imagery have become extremely popular—so popular that almost all traditional film camera manufacturers have switched solely to digital or gone out of business. This meteoric rise has led to an increase in the number of applications that can handle digital imagery. Linux, thanks to its rapid pace of development, is now highly regarded as a multimedia platform for editing digital images.

By default, Ubuntu installs the useful Shotwell Photo Manager. This application is similar to other photo managers, such as iPhoto, and includes simple tools that are adequate for many users, such as red-eye reduction, cropping, color adjustment, and the ability to interact with online photo hosts such as Facebook, Flickr, and Instagram.

The GNU Image Manipulation Program

One of the best graphics clients available is the GNU Image Manipulation Program (GIMP). GIMP is a free, GPL-licensed image editor with sophisticated capabilities that can import and export more than 30 different graphics formats, including files created with Adobe Photoshop. It is often compared with Photoshop, and GIMP represents one of the first significant successes of GNU Project. Many images in Linux were prepared using GIMP.

GIMP is not installed by default, but you can easily install it from the Ubuntu software repositories.

You see an installation dialog box when GIMP is started for the first time, followed by a series of dialog boxes that display information regarding the creation and contents of a local GIMP directory. This directory can contain personal settings, preferences, external application resource files, temporary files, and symbolic links to external software tools used by the editor.

WHAT DOES PHOTOSHOP HAVE THAT GIMP DOES NOT?

Although GIMP is powerful, it does lack two features Adobe Photoshop offers that are important to some graphics professionals.

The first of these is the capability to generate color separations for commercial press printers (CMYK, for the colors cyan, magenta, yellow, and key [or black]). GIMP uses RGB (red, green, and blue), which is great for video display but not so great for printing presses. The second feature GIMP lacks is the use of Pantone colors (a patented color specification) to ensure accurate color matching. These deficiencies might not last long. A CMYK plug-in is in the works (an early version is available from <https://cue.yellow-magic.info/software/sep-arate-plus/index.html>), and the Pantone issues are likely to be addressed in the near future, as well.

If these features are unimportant to you, you will find GIMP to be an excellent tool. If you must use Adobe Photoshop, you might want to explore using Wine or CodeWeavers; there have been consistent reports of success running Photoshop on Linux with these tools. Bear in mind, though, that both Ubuntu and Photoshop release regularly, so check www.winehq.org and www.codeweavers.com for current info before assuming it will work.

Using Scanners in Ubuntu

With the rise of digital photography, there has been an equal decline in the need for image scanners. However, there are still times when you want to use a scanner, and Ubuntu makes it easy with a program installed by default called Simple Scan. Simple Scan is designed to do one thing: scan photos or documents easily. It has few settings or options but does all the things most people would want or need.

You can also use many types of image scanners with GIMP, which is likely to be the choice of people who like to tinker with settings and options or who need greater flexibility than is offered by Simple Scan. If it wasn't installed when you installed GIMP, install the `xsane` package. Then, when you scan from GIMP, you will have an abundance of settings and options that you can use. You can also use XSane by itself.

Working with Graphics Formats

Image file formats are developed to serve a specific technical purpose (lossless compression, for example, where the file size is reduced without sacrificing image quality) or to meet a need for a proprietary format for competitive reasons. Many file formats are covered by one or more patents. For example, at one time the GIF format fell into disfavor with the open source crowd because the patent holder waited a while before deciding to enforce his patent rights rather than being up front with requests for patent royalties. The GIF format is no longer patented, however.

If you want to view or manipulate an image, you need to identify the file format so you can choose the proper tool for working with the image. The file's extension is your first

indicator of the file's format. The graphics image formats supported by the applications included with Ubuntu include the following:

- ▶ **BMP** (.bmp)—Bitmapped graphics, commonly used in Microsoft Windows
- ▶ **GIF** (.gif)—CompuServe Graphics Interchange Format
- ▶ **JPG** (.jpg)—Joint Photographic Experts Group
- ▶ **PCX** (.pcx)—IBM Paintbrush
- ▶ **PNG** (.png)—Portable Network Graphics
- ▶ **SVG** (.svg)—Scalable Vector Graphics
- ▶ **TIF** (.tif)—Tagged Image File format

You can find an extensive list of image file extensions in the man page for ImageMagick, an excellent application included with Ubuntu, which you read more about in upcoming sections of this chapter.

TIP

Ubuntu includes dozens of graphics conversion programs in its software repositories that are accessible through the command line and from a *graphical user interface (GUI)*, and there are few, if any, graphics file formats that cannot be manipulated when using Linux. These programs can be called in Perl scripts, shell scripts, or command-line pipes to support many types of complex format-conversion and image-manipulation tasks. See the man pages for the `ppm`, `pbm`, `pnm`, and `pgm` families of commands. Also see the man page for the `convert` command, which is part of the extremely capable ImageMagick suite of programs.

Sometimes, a file you want to manipulate in some way is in a format that cannot be used by either your graphics application or the final application. The solution is to convert the image file—sometimes through several formats. The `convert` utility from ImageMagick is useful, as is the `netpbm` family of utilities. If it is not already installed, you can easily install ImageMagick from the Ubuntu repositories; the `netpbm` tools are always installed by default.

`convert` is super simple to use from the command line. Here is an example:

```
matthew@seymour~:$ convert image.gif image.png
```

The `convert` utility converts between image formats recognized by ImageMagick. You can also manipulate color depth and size during the conversion process. You can use ImageMagick to append images, surround them with borders, add labels, rotate and shade them, and perform other manipulations well suited to scripting. Other commands associated with ImageMagick include `display`, `animate`, `identify`, and `import`.

The application supports more than 130 different image formats (all listed in the man page for ImageMagick).

The `netpbm` tools are installed by default because they compose the underpinnings of graphics format manipulation. The man page for each image format lists related conversion utilities; the number of those utilities gives you some indication of the way each format is used and shows how one is built on another:

- ▶ **ppm**—The man page for `.ppm`, the *portable pixmap* file format, lists 47 conversion utilities related to `.ppm`. This makes sense because `ppm` is considered the lowest common denominator for color image files. It is therefore often used as an intermediate format.
- ▶ **pgm**—The man page for `.pgm`, the *portable graymap* file format, lists 22 conversion utilities. This makes sense because `.pgm` is the lowest common denominator for grayscale image files.
- ▶ **pnm**—The man page for `.pnm`, the *portable anymap* file format, lists 31 conversion utilities related to it. However, there is no format associated with `.pnm` because it operates in concert with `.ppm`, `.pgm`, and `.pbm`.
- ▶ **pbm**—An examination of the man page for `.pbm`, the *portable bitmap* file format, reveals no conversion utilities. It's a monochrome format and serves as the foundation of the other related formats.

The easiest way to resize or rotate image files is to install the `nautilus-image-converter` package from the repositories. This enables you to right-click an image when you are viewing files in the File Browser (for example, from Places, Pictures) and choose menu options to resize or rotate one or multiple images without opening another program.

Capturing Screen Images

You can use graphics-manipulation tools to capture images that are displayed on your computer screen. Although this technique was used for the production of this book, it has broader uses; there is truth to the cliché that a picture is worth a thousand words. Sometimes showing an example is easier than describing it.

You can use a captured screen image (also called a screen grab or a screenshot) to illustrate an error in the display of an application (a font problem, for example) or an error message that is too complex to copy down by hand. You might just want to share an image of your beautifully crafted custom desktop configuration with your friends or illustrate your written documents.

When using the default desktop, you can take advantage of the built-in screenshot mechanism (`gnome-screenshot`). You can use this tool by pressing the Print Screen key. (Alt+Print Screen takes a screenshot of only the window that has focus on a desktop.) Captured images are saved in PNG format.

Other Graphics Manipulation Options

If you have very specific requirements for working with graphics, you may find one of the following options better suits your needs than the preceding general options and comments. Some, but not all, of these are in the Ubuntu repositories:

- ▶ **Blender**—A 3-D image and animation editor that you can find at www.blender.org
- ▶ **CinePaint**—A powerful and complex tool used by many Hollywood studios that you can find at www.cinepaint.org
- ▶ **darktable**—A RAW editor that can be found at www.darktable.org
- ▶ **digikam**—Photo management software that can be found at www.digikam.org
- ▶ **Hugin**—A panoramic photo stitcher that can be found at <https://hugin.sourceforge.net>
- ▶ **Inkscape**—A vector graphics creation and editing tool that you can find at <https://inkscape.org>
- ▶ **Krita**—A raster graphics creation and editing tool that you can find at <https://krita.org>
- ▶ **POV-Ray**—A powerful and complex 3D graphics program that uses ray tracing and can be found at www.povray.org
- ▶ **Radiance**—Intended for the analysis and visualization of lighting in design and can be found at www.radiance-online.org
- ▶ **Xara Xtreme**—A general-purpose graphics editor that you can find at www.xaraxtreme.org

Using Digital Cameras with Ubuntu

Most digital cameras used with Ubuntu fall into one of two categories: webcams or digital cameras and phone cameras. Ubuntu supports both types. Other types of cameras, such as surveillance cameras that connect directly to a network via wired or wireless connections, need no special support (other than a network connection and viewing software) to be used with a Linux computer.

Ubuntu supports hundreds of different digital cameras, from early parallel-port (CPiA chipset-based) cameras to today's USB-based cameras. You can even use Intel's QX3 USB microscope with Ubuntu.

Handheld Digital Cameras

Because of the good development carried out in the Linux world, you can plug almost any digital camera or camera phone in to your computer through a USB interface, and Ubuntu automatically recognizes the camera as a USB mass storage device. You can even set Ubuntu to recognize when a camera is plugged in so that it automatically imports your photographs for you.

Using Shotwell Photo Manager

Shotwell Photo Manager, mentioned earlier in this chapter, includes simple adjustment tools. You can import your photos into Shotwell, assign tags to them, sort and arrange them, and even upload them to your favorite Internet photo-hosting sites such as Instagram.

Burning CDs and DVDs in Ubuntu

Linux is generally distributed via the Internet as disc images called ISOs that are ready to be written to CDs or DVDs. Therefore, learning how to burn discs is essential if you have to download and install a Linux distribution. You can use CDs and DVDs to do the following:

- ▶ Record and store multimedia data, such as backup files, graphics images, and music.
- ▶ Rip audio tracks from music CDs and compile your own music CDs for your personal use. (*Ripping* refers to extracting music tracks from a music CD.)

Although USB storage devices such as thumb drives are making CDs and DVDs almost as rare as floppy disks, they aren't quite gone, and many people still find them useful. As long as that remains true, we want to make sure this information is available.

Creating CDs and DVDs with Brasero

Although adequate for quick burns and use in shell scripting, the command-line technique for burning CDs and DVDs is an awkward choice for many people (but we still cover it later in this chapter because others find it useful and desirable). Fortunately, Ubuntu provides several graphical clients; the most useful is Brasero.

Brasero is an easy-to-use graphical CD and DVD burning application that is installed by default. Brasero takes a project-based approach to disc burning, opening up with a wizard that offers four different tasks that people commonly want to do. Brasero also remembers previous “projects,” enabling you to quickly create several copies of a disc, which is ideal if you're planning to pass on copies of Ubuntu to your friends and family.

Burning a data CD or DVD is as easy as selecting the option in the opening screen and dragging and dropping the files you want to include from the directory tree on the left to the drop area on the right. If you insert a blank CD or DVD in your writer, Brasero keeps an eye on the disc size and tells you when you reach or exceed the limits. It also creates ISO files, which are disc images that contain everything that would exist on the medium if you burned a real CD or DVD in one file that can be mounted by computer file systems, which is useful if you want to create multiple copies of the same disc or if you want to share a disc image, perhaps using a USB thumb drive or over the Internet.

Finally, click the Burn button, input a label for the disc, and Brasero starts creating your new CD or DVD or image file. How long it takes to create a CD or DVD depends on the amount of data you are writing and the speed of your drive.

Creating CDs from the Command Line

In Linux, creating a CD at the command line is a two-step process. You first create the ISO9660-formatted image, and then you burn or write the image onto the CD. The ISO9660 is the default file system for CD-ROMs.

Use the `mkisofs` command to create the ISO image. The `mkisofs` command has many options (see the man page for a full listing), and you use the following for quick burns:

```
matthew@seymour~:$ mkisofs -r -v -J -l -o /tmp/our_special_cd.iso /
source_directory
```

The options used in this example are as follows:

- ▶ `-r`—Sets the permission of the files to more useful values. `UID` and `GID` (individual and group user ID requirements) are set to `0`, all files are globally readable and searchable, and all files are set as executable (for Windows systems).
- ▶ `-v`—Displays verbose messages (rather than terse messages) so that you can see what is occurring during the process; these messages can help you resolve any problems that occur.
- ▶ `-J`—Uses the Joliet extensions to ISO9660 so that your Windows-using buddies can more easily read the CD. The Joliet (for Windows), Rock Ridge (for UNIX), and HSF (for Mac) extensions to the ISO9660 standard are used to accommodate long filenames rather than the eight-character DOS filenames that the ISO9660 standard supports.
- ▶ `-l`—Allows 31-character filenames; DOS does not like it, but everyone else does.
- ▶ `-o`—Defines the directory where the image will be written (that is, the output) and its name. The `/tmp` directory is convenient for this purpose, but the image could go anywhere you have write permissions.
- ▶ `/source_directory`—Indicates the path to the source directory—that is, the directory containing the files you want to include. There are ways to append additional paths and exclude directories (and files) under the specified path; it is all explained in the man page, so check there if you need that level of complexity. The simple solution is to construct a new directory tree and populate it with the files you want to copy and then make the image using that directory as the source.

Many more options are available, including options to make the CD bootable.

After you have created the ISO image, you can write it to the CD with the `cdrecord` command:

```
matthew@seymour~:$ sudo cdrecord -eject -v speed=12 dev=0,0,0 /tmp/
our_special_cd.iso
```

The options used in this example are as follows:

- ▶ `-eject`—Ejects the CD when the write operation is finished.
- ▶ `-v`—Displays verbose messages.

- ▶ **speed=**—Sets the speed; the rate depends on the individual drive’s capabilities. If the drive or the recordable medium is poor, you can use lower speeds to get a good burn.
- ▶ **dev=**—Specifies the device number of the CD writer.

NOTE

You can also use the `blank = option` with the `cdrecord` command to erase CD-RW discs. The `cdrecord` command has fewer options than `mkisofs` does, but it offers the `-multi` option, which enables you to make multisession CDs. A multisession CD enables you to write a data track, quit, and then add more data to the CD later. A single-session CD can be written to only once; any leftover CD capacity is wasted. Read about other options in the `cdrecord` man page.

Current capacity for CD media is 700MB of data or 80 minutes of music. (There are 800MB/90-minute CDs, but they are rare.) Some CDs can be overburned—that is, recorded to a capacity in excess of the standard. The `cdrecord` command and some graphical programs are capable of overburning if your CD-RW drive supports it. You can learn more about overburning CDs at www.cdmediaworld.com/hardware/cdrom/cd_oversize.shtml/.

Creating DVDs from the Command Line

There are several competing formats for DVD, as follows:

- ▶ DVD+R
- ▶ DVD-R
- ▶ DVD+RW
- ▶ DVD-RW

Differences in the + and - formats mostly have to do with how the data is modulated onto the DVD itself, with the + format having an edge in buffer underrun recovery. How this is achieved affects the playability of the newly created DVD on any DVD player. The DVD+ format also has some advantages in recording on scratched or dirty media. Most drives support the DVD+ format. As with any other technology, your mileage may vary.

We focus on the DVD+RW drives because most drives support that standard. The software supplied with Ubuntu has support for writing to DVD-R/W (rewritable) media, as well. It will be useful for you to review the DVD+RW/+R/-R[W] for Linux HOWTO at <https://fy.chalmers.se/~appro/linux/DVD+RW/> before you attempt to use `dvd+rw-tools`, which you need to install to enable DVD creation (also known as *mastering*) and the `cdrttools` package. You can ignore the discussion in the HOWTO about kernel patches and compiling the tools.

TIP

The 4.7GB size of DVD media is measured as 1,000 megabytes per gigabyte instead of the more traditionally used, but not entirely accurate, 1,024 megabytes per gigabyte (more appropriately written GiB), so do not be surprised when the actual formatted capacity, about 4.4GB, is less than you anticipated. Most hard drive manufacturers have also made the switch. `dvd+rw-tools` does not allow you to exceed the capacity of the disc.

You need to have the `dvd+rw-tools` package installed, as well as the `cdrtools` package. The `dvd+rw-tools` package contains the `growisofs` application (which acts as a front end to `mkisofs`) and the DVD formatting utility.

You can use DVD media to record data in two ways. The first way is much the same as that used to record CDs in a session, and the second way is to record the data as a true file system, using packet writing.

Session Writing

To record data in a session, you use a two-phase process:

1. Format the disc with `dvd+rw-format /dev/scd0` (which is necessary only the first time you use a disc, where `/dev/scd0` is the device name for your drive).
2. Write your data to the disc with `growisofs -Z /dev/scd0 -R -J /your_files`.

The `growisofs` command simply streams the data to the disc. For subsequent sessions, use the `-M` argument rather than `-z`. The `-z` argument is used only for the initial session recording; if you use the `-z` argument on an already-used disc, it erases the existing files.

CAUTION

Some DVDs come preformatted; formatting them again when you use them for the first time can make the DVDs useless. Always be sure to carefully read the packaging your DVD comes in to ensure that you are not about to create another coaster.

TIP

Writing a first session of at least 1GB helps maintain compatibility of your recorded data with other optical drives. DVD players calibrate themselves by attempting to read from specific locations on the disc; you need data there for the drive to read it and calibrate itself.

Also, because of limitations to the ISO9660 file system in Linux, do not start new sessions of a multisession DVD that would create a directory past the 4GB boundary. If you do so, it causes the offsets used to point to the files to “wrap around” and point to the wrong files.

Packet Writing

Packet writing treats the CD or DVD like a hard drive, in which you create a file system (like `ext3`) and format the disc and then write to it randomly as you would to a conventional hard drive. This method, although commonly available on Windows-based

computers, was long considered experimental for Linux and was never used much anyway because USB thumb drives became common before the use of CD or DVD-RWs had the opportunity. We do not cover this in detail here, but a quick overview is appropriate.

TIP

DVD+RW media are capable of only about 1,000 writes, so it is useful to mount them with the `noatime` option to eliminate any writing to update their inodes or simply mount them as read-only when it's not necessary to write to them.

Piping data to the `growisofs` command is possible:

```
matthew@seymour~:$ sudo your_application | growisofs -Z /dev/scd0=/dev/fd/0
```

Burning from an existing image (or file, or named pipe, or device) is also possible:

```
matthew@seymour~:$ sudo growisofs -Z /dev/scd0=image
```

The `dvd+rw-tools` documentation, found at `/usr/share/doc/dvd+rw-tools/index.html`, is required reading before your first use of the program. We also suggest that you experiment with DVD-RW (rewritable) media first because if you make mistakes, you can still reuse the discs instead of creating several new coasters for your coffee mug.

Viewing Video

You can use Ubuntu tools and applications to view videos.

Video Formats

Ubuntu recognizes a variety of video formats. The formats created by the MPEG group, Apple, and Microsoft dominate, however. At the heart of video formats are the *codecs*—the encoders and decoders of video and audio information. Codecs are typically proprietary, but free codecs do exist. Here is a list of the most common video formats and their associated file extensions, although many more exist:

- ▶ **AVI (.avi)**—The Windows audiovisual format
- ▶ **FLV (.flv)**—Used in Adobe Flash; supports H.264 and others
- ▶ **MPEG (.mpeg)**—The MPEG video format; also known as .mpg
- ▶ **MOV (.mov)**—A QuickTime video format
- ▶ **OGV/OGG (.ogv/.ogg)**—The Ogg Theora freely licensed video format
- ▶ **QT (.qt)**—The QuickTime video format from Apple
- ▶ **WEBM (.webm)**—Google's royalty-free container for audio and video (such as in VP8 format) designed for HTML5

Viewing Video in Linux

Out of the box, Ubuntu does not support any of the proprietary video codecs due to licensing restrictions. However, this functionality can be acquired if you install the `ubuntu-restricted-extras` package from the Ubuntu software repositories. You can learn more about it at <https://help.ubuntu.com/community/RestrictedFormats>.

You can watch video files and video DVDs with Totem Movie Player, which is installed by default. This may also be used with several other file formats and for both video and audio and is especially well suited for almost anything you are likely to want to use after you install the `ubuntu-restricted-extras` package.

A more powerful video viewer application is VLC, which is available in the software repositories and also for other operating systems, like Windows and macOS. VLC uses its own set of audio and video codecs and supports a wider range of video formats than any other media player we have encountered. If VLC can't play it, it probably can't be played.

Recording and Editing Audio

Recording and editing audio files in Ubuntu is possible using a number of applications. For professional-grade production, you will want a digital audio workstation (DAW) like Ardour or LMMS. For most of us, something simpler like Audacity will suffice and still produce excellent results. These are often used to record and edit audio tracks that are then imported into one of the video editors described in the following section.

These are the most respected audio recording and editing applications for Linux:

- ▶ **Audacity**—This cross-platform (Linux, macOS, and Windows) audio editor for Linux is one of the most commonly used and can be found at www.audacityteam.org/.
- ▶ **Ardour**—This cross-platform (Linux, macOS, and Windows) DAW app is used by many professionals and can be found at <https://ardour.org/>.
- ▶ **Cecilia**—This cross-platform (Linux, macOS, and Windows) audio signal processing software is aimed at sound designers and can be found at <https://ajaxsoundstudio.com/software/cecilia/>.
- ▶ **LMMS**—Also cross-platform (Linux, macOS, and Windows) this DAW is made and maintained by musicians; find it at <https://lmms.io/>.
- ▶ **Mixxx**—This virtual DJ tool can be found at www.mixxx.org/.
- ▶ **Rosegarden**—Too simple to be called a DAW, but not entirely basic, this audio editor might be appreciated by casual users and can be found at www.rosegardenmusic.com/.

Most of these are available in the Ubuntu repositories, so check there before you install from the website.

Editing Video

Recording and editing video in Ubuntu is possible using a number of applications. For professional-grade production, you will want something like Blender. For most of us, something simpler like PiTiVi will suffice and still produce excellent results.

These are the most respected video editing applications for Linux:

- ▶ **Avidemux**—This tool designed for people with simple needs can be found at <https://fixounet.free.fr/avidemux/>.
- ▶ **Blender**—This cross-platform (Linux, macOS, and Windows) professional-grade video production package can be found at www.blender.org.
- ▶ **Cinelerra**—This tool has been around for years but recently made some big changes and is rebuilding its community. Find it at <https://cinelerra.org>.
- ▶ **DaVinci Resolve**—This cross-platform (Linux, macOS, and Windows) professional-grade video production package is commonly used in Hollywood and can be found at www.blackmagicdesign.com/products/davinciresolve/.
- ▶ **Kdenlive**—This comes from the KDE folks and can be found at <https://kdenlive.org>.
- ▶ **Lightworks**—This cross-platform (Linux, macOS, and Windows) professional-grade video production package can be found at www.lwks.com/.
- ▶ **OpenShot Video Editor**—This editor can be found at www.openshot.org.
- ▶ **PiTiVi**—Find this tool at www.pitivi.org/.
- ▶ **Shotcut**—This cross-platform (Linux, macOS, and Windows) editor can be found at <https://shotcut.org/>.

Most of these options are in the Ubuntu repositories, so check there first, but some you have to download directly from their websites. Not all of them are free software.

References

- ▶ **www.videolan.org**—A multimedia player project with good documentation that will play almost anything
- ▶ **www.gimp.org**—Home page of GIMP (*GNU Image Manipulation Program*)
- ▶ **www.sane-project.org**—Home page of the SANE (*Scanner Access Now Easy*) project
- ▶ **www.imagemagick.org**—Home page for ImageMagick
- ▶ **<https://gimp.net/tutorials/>**—Official tutorials for GIMP

This page intentionally left blank

CHAPTER 8

Games

Playing games is a fun part of computing. For some of us, games are a big part of the appeal of any operating system, at least for uses beyond the business or corporate environment. From the humble card games that once entertained millions during their coffee breaks to the heavily involved first-person shooters that involve players dotted around the globe, Linux offers a quality gaming platform that might surprise you.

In this chapter, we explore some of the common games available for you to download and install easily in Ubuntu.

Ubuntu Gaming

A small number of games come installed by default with standard desktop Ubuntu, mostly simple games to divert your attention for a few minutes between tasks, such as *Solitaire* and *Mahjongg*. These are easy to try out and learn, and you might find them enjoyable.

Most of us who enjoy games will want to find something more. Thankfully, games for Linux do not stop there; several popular games have native Linux support. We discuss a small number of our favorites here.

Emulators enable you to play classic games, such as the classic LucasArts ScummVM's *Secret of Monkey Island*, natively under Linux. There are emulators for DOS, NES, SNES, and many more platforms. If you are interested in them, search online for DGen/SDL, DOSBox, xtrs, FCE Ultra, GnGeo, SDLmame, ScummVM, and Stella. The documentation for emulators is hit or miss, but if you are lucky, you might be able to play your favorite old games for consoles or operating systems that no longer exist or that current OS versions do not support.

IN THIS CHAPTER

- ▶ Ubuntu Gaming
- ▶ Installing Proprietary Video Drivers
- ▶ Online Game Sources
- ▶ Installing Games from the Ubuntu Repositories
- ▶ Playing Windows Games
- ▶ References

More importantly for most of us, a large number of current releases of great games are available for Linux.

First, if we have an AMD or NVIDIA video card, we need to upgrade the default video drivers to the proprietary drivers that have greater capabilities.

Installing Proprietary Video Drivers

A major gripe of Linux users has been the difficulty involved in getting modern 3D graphics cards to work. Thankfully, both AMD and NVIDIA support Linux, albeit NVIDIA does so by using proprietary drivers. Ubuntu will install on either type of system, but with NVIDIA cards it will use the open source Nouveau driver, which works for most things but does not include all the functionality of the highest-end cards. This means that Ubuntu does not ship with native proprietary drivers activated for either graphics card, but they can easily be installed, often during or just after the operating system is installed. These drivers are needed for most visually spectacular games. The open source AMD drivers work well for gaming, although for some professional use cases like computer-aided design (CAD), a person might want to upgrade to the proprietary AMD drivers available from their website.

Proprietary drivers are hard for some Linux distros to include as part of their standard installation. The Ubuntu community has taken a pragmatic approach of including the NVIDIA drivers within the main Ubuntu distro, but they're disabled by default. Therefore, anyone who has NVIDIA hardware can activate those drivers to take advantage of their features.

NOTE

If you think that using proprietary drivers is the only way on Linux, we should mention that a lot of development is going into providing totally free and open source drivers for slightly older graphics cards. Ubuntu automatically selects the best “free” driver for your system and allows you to switch the proprietary driver if you want to. Although the open source drivers provide 3D acceleration, this support doesn't always extend to a full feature set or to more recent graphics cards.

Activating the proprietary driver if you need to is easy: Open Software Updater, click Settings, and then select the Additional Drivers tab, as shown in Figure 8.1.

Installing a different hardware driver requires super user privileges, so you are asked for your password. When the drivers list appear, read the descriptions and look for the recommended entry for your hardware, highlight it, and click Apply Changes at the lower right. Ubuntu confirms that you want to use the proprietary driver and, if you agree, automatically downloads and configures the relevant driver. If you want to revert to the open driver, do the same thing but select the activated driver and click Remove at the lower right, in the same place where the previously mentioned Apply Changes button appeared.

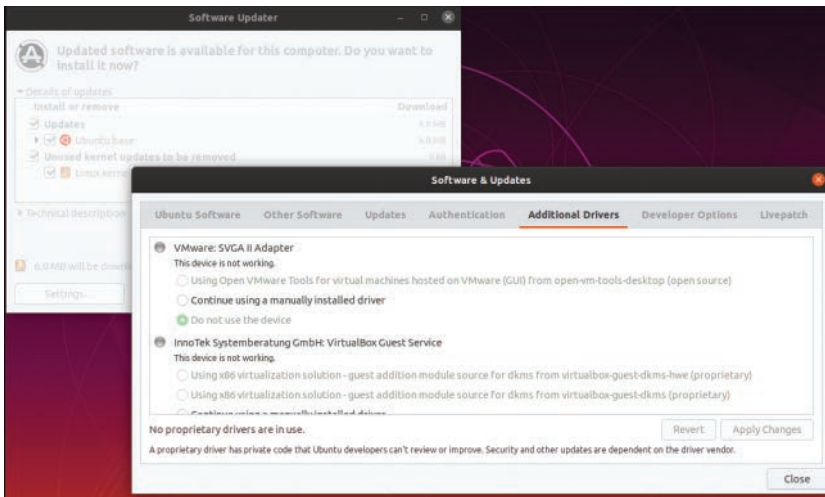


FIGURE 8.1 Use the Additional Drivers tab to activate or deactivate the appropriate proprietary graphics driver for your graphics card.

Online Game Sources

Many websites provide downloadable games, typically for pay, although some titles are free. The following sites all have games available that will run on Ubuntu, as of this writing.

Steam

Steam is a cross-platform entertainment platform. When you install it on Ubuntu, you get access to a ton of games and other fun things. Some of them are free, but most require you to pay.

Using Steam has become the primary method that most current Linux-based gamers use to find, install, and play games.

Steam has grown into the premier source for quality professional games on Linux and for a lot of indie content as well. Steam is created by Valve Software, usually referred to as just “Valve,” a well-established gaming company with a great reputation because of games like *Half-Life*.

In 2013, Gabe Newell, the cofounder and managing director of Valve, said that Linux is the future of gaming, despite its current minuscule share of the market, and the Steam website proclaims Ubuntu as Steam’s favorite version of Linux. Steam is Newell’s company’s means of pursuing that future.

To learn more about Valve and Steam, see www.valvesoftware.com and <https://store.steampowered.com>.

GOG.com

GOG.com offers a collection of thousands of games that run on multiple platforms, including Linux. The games are all DRM-free, which means that no digital rights management is included, making free and open source (FOSS) proponents happy. Take a look at www.gog.com/games.

Humble

The Humble store has a number of FOSS games for sale. The site is known for its bundles, selling games for a much-cheaper-than-usual price when you buy them in a package. To see options available, visit www.humblebundle.com/store.

itch.io

This cross-platform site offers many Linux-compatible games. Visit <https://itch.io/games/platform-linux>.

LGDB

The Linux Game Database (LGDB) has more than 2,000 games available to download, along with emulators, tools, and a web forum to discuss games, gaming, and Q&A with other users. You can find the games list at <https://lgdb.org/games>.

Game Jolt

The games on this site are all indie games, meaning they were created by individuals or small companies. Like the other sites, this site offers many cross-platform games. You can find a list of the Linux options at <https://gamejolt.com/games/best?os=linux>.

Installing Games from the Ubuntu Repositories

In this section, you learn how to install some of the popular games for Ubuntu, all of which can be easily installed from Ubuntu software repositories. Alongside the usual shoot-'em-up games, you'll also find one or two strategy-focused titles.

Warsow

Warsow is a free and fast-paced *first-person shooter (FPS)* game that is available on Windows, macOS, and Linux. Members of the Ubuntu community have packaged the game and made it available directly and easily for Ubuntu users from the software repositories. The game involves moving quickly, grabbing power-ups and weapons before your enemies do, and trying to plant a bomb and steal your enemy's flag without anyone seeing you. You can jump, dash, dodge, and even wall jump your way throughout the colorful 3D environment. Figure 8.2 shows a game just getting started, with lots of weapons, platforms and ramps, and power-ups in sight. Many other FPS games are available, such as *Alien Arena*, *Urban Terror*, *Wolfenstein: Enemy Territory*, *Smokin' Guns*, *Nexuiz*, *World of Padman*, and *Cube 2: Sauerbraten*.

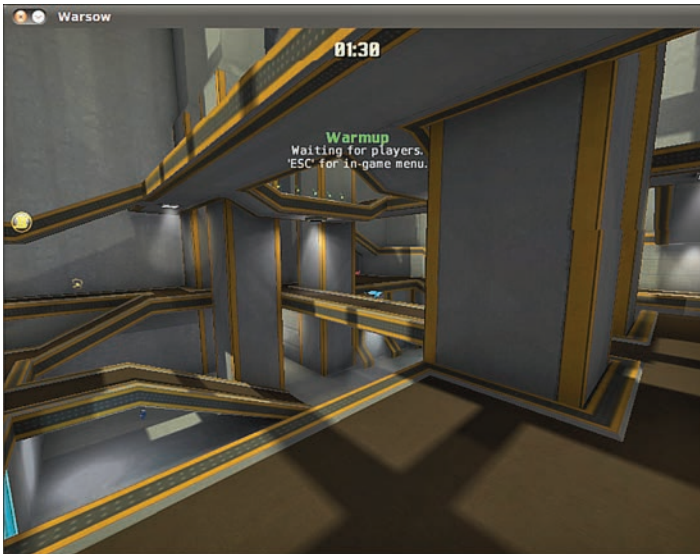


FIGURE 8.2 *Warsow* is one of the newest and fastest FPS games available.

Scorched 3D

Scorched 3D is based on an old DOS game called *Scorched Earth*. The object and game play are similar: There are multiple players, and you enter targeting details to try to destroy the other players using a variety of missile-like weapons. You earn points for each win and can buy new weapons at the end of each round. The game offers an amazing array of weapons, and the game play is completely in stunning 3D, as shown in Figure 8.3.



FIGURE 8.3 *Scorched 3D* in action.

Scorched 3D is based on turns. Each player shoots once, and then each of the other players takes a shot before the first player shoots again. The game allows you to have more than 20 players at the same time, including both human and computer-controlled players. You may play a local game, over a LAN, or even over the Internet. *Scorched 3D* runs on Windows, macOS, and Linux, so you may play with your friends regardless of what platforms their computers use.

Frozen Bubble

Frozen Bubble is an amusing little game with sharp graphics, nice music, and easy game play. You may play alone or against another player. The goal is to use your colored marbles to knock down the ones above you within a certain amount of time. You have to hit at least two at a time that are the same color as the one you shoot for anything to fall; otherwise, your shot joins the menagerie above and brings about your demise somewhat more quickly, as illustrated in Figure 8.4. There is a lot more to *Frozen Bubble*, but even with all the details, it is easy enough for a child to play and interesting enough to hold the interest of most adults.



FIGURE 8.4 Move left or right to aim and press up to fire.

SuperTux

Many of us grew up in the era when game play was more important than graphics. Even so, we still liked flashy and pretty-looking games. *SuperTux* is a throwback to the Mario era. It is a 2D side-scroller in which you jump, run, and occasionally shoot something if you have the appropriate power-up. The hero is Tux, the Linux penguin. More than

25 levels are available, as is a level editor for you to create your own. If you enjoy running, jumping, hitting your head to get money, and jumping on your enemies, this game is for you. Figure 8.5 gives you the basic look and feel.



FIGURE 8.5 Although the look and feel of *SuperTux* takes you back to the Mario era, it is far from dull or boring.

Battle for Wesnoth

One of the most popular games currently available for Linux is *Battle for Wesnoth* (see Figure 8.6), a strategy game featuring both single-player and multiplayer options. Based in a fantasy land, you are responsible for building armies to wage war against foes who are attacking you. Game play may be based on scenarios, such as in single-player mode, where some scenarios are preinstalled and others may be easily downloaded, or based on trying to better your friends at a LAN party or online.

Battle for Wesnoth comes with a map editor that lets you create your own scenarios. An active community shares their work and welcomes new contributions. You can find more information about *Battle for Wesnoth* at <https://wesnoth.org>.

Frets on Fire

Frets on Fire, shown in Figure 8.7, is similar to *Guitar Hero*. Players try to keep up with a song and “play” it correctly. The game supports songs from *Guitar Hero I* and *Guitar Hero II* but, unlike those proprietary games, is expandable by the community, as well, with more songs available for download from the Internet. *Frets on Fire* is completely open source and has content-compatible versions for Linux, Windows, and macOS.



FIGURE 8.6 Flex your strategic brain by playing *Battle for Wesnoth*, a rich and full land of fantasy of adventure.



FIGURE 8.7 *Frets on Fire* offers flashy graphics and familiar game play.

FlightGear

If you like flight simulators, you should try out *FlightGear* (see Figure 8.8). It is cross-platform, sophisticated, and fun. It is completely open source and developed by volunteers, and it's also very professional looking and smooth. If you like this, you can also try other similar games, such as *Thunder and Lightning*, *GL-117*, and *Search and Rescue II*.



FIGURE 8.8 *FlightGear* features stunning landscapes and technically accurate control panels.

Speed Dreams

Many racing games are available. One of the newer ones is *Speed Dreams* (see Figure 8.9), which started as a fork of an older racing game, *TORCS*. It features a variety of tracks, vehicles, and options. *Speed Dreams* is open source, free, and multiplatform. Similar games include *TORCS*, *Rigs of Rods*, *vDrift*, *Tile Racer*, *Trigger*, and *Mania Drive*.



FIGURE 8.9 In *Speed Dreams*, many views of the race are available, including one from your car's hood.

Games for Kids

Kids, both young and old, have Linux game options. Check out game packages such as *gCompris*, *Childsplay*, and *Tux Paint* to get started. Some of these are educational, some teach computer skills such as using a mouse or a keyboard, and others are just for amusement. Many developers are also parents who have placed a high priority on making quality software for their children to enjoy on their favorite platform. You can search for children's games in the Ubuntu Software Center and find lots of great things to try.

Commercial Games

In the past few years, Ubuntu and Canonical have made a way for commercial software companies to make their products—including games—available for installation in Ubuntu via the Ubuntu Software Center. A special section of the Ubuntu Software Center labeled For Purchase has been created. Look there for some new and flashy options from commercial vendors. Payment is required, and the details are taken care of within the Ubuntu Software Center.

One interesting and cool commercial game option is the Humble Indie Bundle from www.humblebundle.com, which is a collection of cross-platform software made by independent developers with a “pay what you can” price tag. These bundles are made available intermittently, and sometimes you can find them available directly from the Ubuntu Software Center. This is a great way to find some fun games with a low cost while feeling good about supporting people who write software for fun.

Playing Windows Games

Ubuntu is primarily aimed at desktop computer users who want a system that is stable, powerful, and easy to use. It is not primarily a gaming platform. In fact, compared to Windows, there are not nearly as many games available or being developed for Linux (although the number is growing and has improved). This doesn't mean hard-core gamers need to feel left out, though. Two projects exist to help game lovers play most Windows-based games on Linux.

A project called Wine uses application interfaces to make Windows programs believe they are running on a Windows platform and not a Linux platform. Bear in mind that Wine, which is a compatibility layer, stands for *Wine Is Not an Emulator*, so you should not think of it as one; the community can get quite touchy about it! Although the open source, free software project Wine won't run everything, it does run a very large number of Windows programs, including many games.

Crossover Games is another commercial option available in the Ubuntu Software Center under For Purchase.

TIP

The keys to successful gaming in Linux are to always read the documentation thoroughly, always investigate the Internet resources thoroughly, and always understand your system. Installing games is a great way to learn about your system because the reward of success is so much fun.

References

- ▶ **www.warsow.net**—The official site of *Warsow*
- ▶ **www.scorched3d.co.uk**—The official site of *Scorched 3D*
- ▶ **www.frozen-bubble.org**—The official site of *Frozen Bubble*
- ▶ **https://supertux.lethargik.org**—The official site of *SuperTux*
- ▶ **https://wesnoth.org**—The official site of *Battle for Wesnoth*
- ▶ **https://gcompris.net**—The official site of *gCompris*
- ▶ **https://fretsonfire.sourceforge.net**—The official site of *Frets on Fire*
- ▶ **https://childsplay.sourceforge.net/index.php**—The official site of *Childsplay*
- ▶ **www.tuxpaint.org**—The official site of *Tux Paint*
- ▶ **www.flightgear.org**—The official site of *FlightGear*
- ▶ **www.speed-dreams.org**—The official site of *Speed Dreams*
- ▶ **https://help.ubuntu.com/community/Games**—Ubuntu community documentation for playing games on Ubuntu
- ▶ **www.nvidia.com/object/unix.html**—The home page for NVIDIA UNIX/Linux drivers
- ▶ **https://help.ubuntu.com/community/Wine**—Ubuntu community documentation for Wine
- ▶ **www.winehq.org**—The official site of Wine, which includes good information about software that is known to work with the current version in an application database subsite at <https://appdb.winehq.org>

This page intentionally left blank

CHAPTER 9

Managing Software

In this chapter, we look at the options for managing your software in Ubuntu. If you are used to an environment where you are reliant on visiting different vendor websites to download updates, you are in for a pleasant surprise. Updating a full Ubuntu installation, including all the application software, is as simple as running the Update Manager program. You will discover just how easy it is to install and even remove various software packages.

Ubuntu provides a variety of tools for system resource management. The following sections introduce the graphical software management tools you will use for most of your software management. This chapter also covers monitoring and managing memory and disk storage on your system.

Ubuntu Software

Ubuntu Software is a graphical utility for package management in Ubuntu. You can find it by searching Activities for “Ubuntu Software”; the package and executable program is named `ubuntu-software`. Ubuntu Software enables you to easily select and install a large array of applications by using the intuitive built-in search and easy one-click installation. When you open the program, you see the main screen, as shown in Figure 9.1.

Along the top side of the screen, you have three menu options: All, Installed, and Updates. To the right is a search icon you can use to search for packages. Scroll down to find software listed by categories.

Installing new software via Ubuntu Software is as simple as finding the software in the package list, double-clicking it, and then clicking the Install button. When you do so, you may be asked for your password; then the application is downloaded and installed. You can remove an application by finding it in Ubuntu Software and clicking the Remove button.

IN THIS CHAPTER

- ▶ Ubuntu Software
- ▶ Using Synaptic for Software Management
- ▶ Staying Up to Date
- ▶ Working on the Command Line
- ▶ Compiling Software from Source
- ▶ Configuration Management
- ▶ Using the Snappy Package Manager
- ▶ References

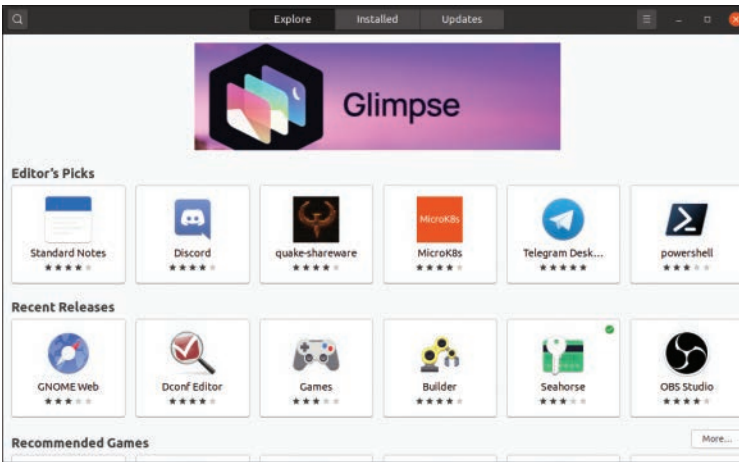


FIGURE 9.1 The initial Ubuntu Software screen enables you to browse through packages sorted into categories.

Using Synaptic for Software Management

Ubuntu Software works just fine for adding and removing applications, but if you need to install something specific—such as a library—you need to use the command line or Synaptic (see Figure 9.2). You can install Synaptic by using Ubuntu Software; it is not installed by default.

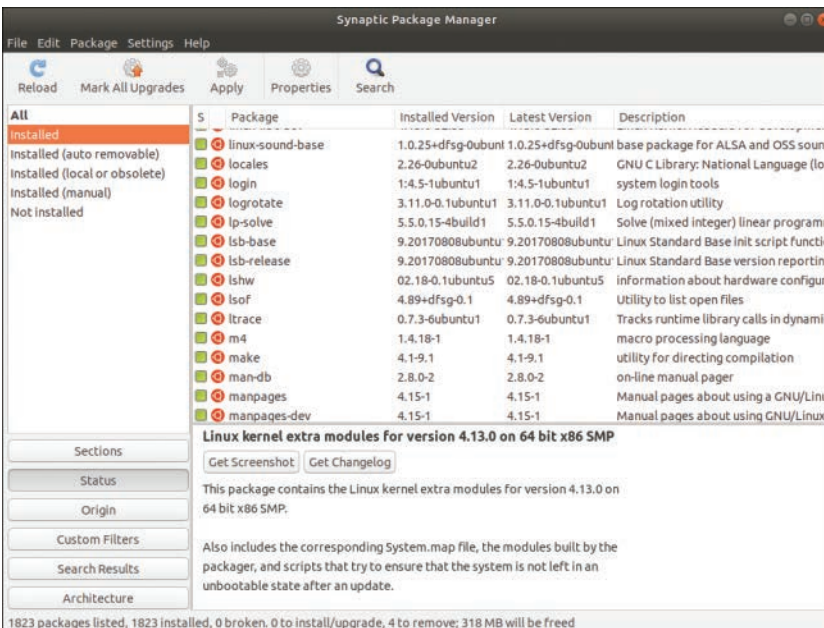


FIGURE 9.2 For more advanced software management in a GUI, Synaptic is the preferred tool.

Along the left are software categories, along the top right are the package selections for that category, and on the bottom right is the Package Information window, which shows information about the currently selected package. To install or remove software, click the check box to the left of its name, and you see a menu that offers the following options:

- ▶ **Unmark**—If you have marked the package for installation, upgrade, or one of the other options, this option removes that mark.
- ▶ **Mark for Installation**—Click this option to add this package to the list that will be installed.
- ▶ **Mark for Re-installation**—If you have some software already installed, but for some reason it's not working, click this option to reinstall it from scratch. Existing configuration files are retained, so any edits you have made are safe.
- ▶ **Mark for Upgrade**—If the software has updates available, select this option to download and install them.
- ▶ **Mark for Removal**—Select this option to delete the selected package from your system but leave its configuration files intact so that if you ever reinstall it, you do not have to reconfigure it.
- ▶ **Mark for Complete Removal**—Select this option to delete the selected package from your system and also remove any configuration files and purge everything from the system.

After you have made your changes, click the Apply button to have Synaptic download, install, upgrade, and uninstall as necessary. If you close the program without clicking Apply, your changes are lost.

Beneath the categories on the left side of the screen are six buttons: Sections, Status, Origin, Custom Filters, Search Results, and Architecture. These customize the left list: Sections is the Categories view; Status enables you to view packages that are installed or upgradable; Origin lists the different repositories available to download packages; Custom Filters has some esoteric groupings that are useful only to advanced users; Search Results stores results of your searches; and Architecture shows the packages specific to each architecture of Ubuntu.

You can press Ctrl+F at any time to search for a particular package. By default, it is set to search by package description and name. You may change the Look In box setting to only search for name. As mentioned earlier, your search terms are saved under the Search view (the button on the bottom left), and you can click from that list to search again on that term.

As well as providing the method of installing and removing software, Synaptic provides the means to configure the servers you want to use for finding packages. In fact, this is where you can make one of the most important changes to your Ubuntu system: You can open it up to the Ubuntu Universe and Multiverse.

Ubuntu is based on the Debian distribution, which has thousands of software packages available for installation. Ubuntu uses only a subset of that number but makes it easy

for you to install the others, along with many packages that are not available in Debian. When you use Synaptic, you see small orange Ubuntu logos next to many packages, which identify them as being officially supported by the Canonical-supported Ubuntu developers. The packages that do not have this logo are supported by the wider Ubuntu community of developers.

To enable the Universe and Multiverse repositories, go to Settings, Repositories. This list shows all the servers you have configured for software installation and updates and includes the Universe and Multiverse repositories. When you find them, check them, and then click Close.

Synaptic shows a message box, warning you that the repository listings have changed and that you need to click the Reload button (near the top left of the Synaptic window) to have it refresh the package lists. Go ahead and do that, and you should see a lot more software options appear for your selection. However, notice that only some software options have the official Ubuntu seal attached, and for anything that does not have the seal, you may want to be cautious when installing it.

NOTE

Much of the software discussed in this book is available only through the Universe repository. Therefore, we highly recommend enabling it to get full use out of this book and your Ubuntu installation.

Staying Up to Date

Although you can manage your software updates through Synaptic, Ubuntu provides a dedicated tool called Software Updater (shown in Figure 9.3). This tool is designed to be simple to use: When you run it, Software Updater automatically downloads the list of updates available and checks them all in the list it shows. All you need to do is click Install Now to bring your system up to date. If you want a little more information about the updates, click Technical Description to see what has changed in the update.

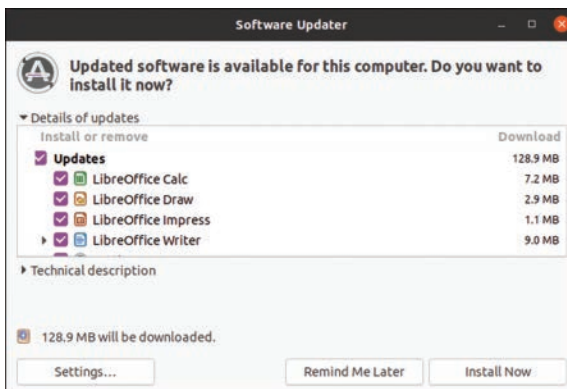


FIGURE 9.3 If you need to update your software to apply bug fixes and security upgrades, use Software Updater.

Ubuntu automatically checks for updates periodically and notifies you when critical updates are available. However, there's no harm in running Software Updater yourself every so often, just to make sure; it's better to be safe than sorry.

Working on the Command Line

With so much software available for installation, it is no surprise that Debian-based distros provide many ways to manage software installation. At their root, however, they all use Debian's world-renowned *Advanced Package Tool (APT)*. A person posting on Slashdot.com once said, "Welcome to Slashdot. If you can't think of anything original, just say how much APT rocks and you'll fit right in." You see, even though many other distros have tried to equal the power of APT, nothing else even comes close.

Why is APT so cool? Well, it was the first system to properly handle dependencies in software. Other distros use files that have dependencies. For example, Red Hat used RPM files that had dependencies. An RPM for GIMP would have a dependency on GTK, the graphical toolkit on which GIMP is based. As a result, if you tried to install your GIMP RPM without having the GTK RPM, your install would fail. So, you grab the GTK RPM and try again. Aha: GTK has a dependency on three other things that you need to download, and those three other things have dependencies on 20 other things...and so on, usually until you can't find a working RPM for a dependency, and you give up.

APT, on the other hand, was designed to automatically find and download dependencies for your packages. So, if you want to install GIMP, APT downloads GIMP's package and any other software it needs to work. No more hunting around by hand, no more worrying about finding the right version, and certainly no more need to compile things by hand. APT also handles installation resuming, which means that if you lose your Internet connection partway through an upgrade (or if your battery runs out, or if you have to quit, or whatever), APT picks up where it left off the next time you run it.

Day-to-Day APT Usage

To enable you to search for packages both quickly and thoroughly, APT uses a local cache of the available packages. Try running this command:

```
matthew@seymour:~$ sudo apt update
```

The `apt update` command instructs APT to contact all the servers it is configured to use and download the latest list of file updates. If your lists are outdated, it takes a minute or two for APT to download the updates. Otherwise, this command executes it in a couple of seconds.

After the latest package information has been downloaded, you are returned to the command line. You can now ask APT to automatically download any software that has been updated, using this command:

```
matthew@seymour:~$ sudo apt upgrade
```

If you have a lot of software installed on your machine, there is a greater chance of things being updated. APT scans your software and compares it to the latest package information from the servers and produces a report something like this:

```
mmatthew@seymour:~$ sudo apt upgrade
Reading package lists... Done
Building dependency tree
Reading state information... Done
The following packages will be upgraded:
  cabextract google-chrome-beta icedtea6-plugin language-pack-en
  language-pack-en-base language-pack-gnome-en language-pack-gnome-en-base
  libfreetype6 libfreetype6-dev libsmbclient libwbclient0 openjdk-6-jre
  openjdk-6-jre-headless openjdk-6-jre-lib samba-common samba-common-bin
  smbclient upstart winbind xserver-common xserver-xorg-core
21 upgraded, 0 newly installed, 0 to remove and 0 not upgraded.
Need to get 84.8MB of archives.
After this operation, 623kB of additional disk space will be used.
Do you want to continue [Y/n]?
```

Each part of this report tells you something important. Starting at the top, the line, “The following packages will be upgraded,” gives you the exact list of packages for which updates are available. If you’re installing new software or removing software, you see lists titled, “The following packages will be installed” and “The following packages will be removed.” A summary at the end shows a total of 21 packages that APT will upgrade, with 0 new packages, 0 to remove, and 0 not upgraded. Because this is an upgrade rather than an installation of new software, all those new packages take up only 623KB of additional space. Although you have an 84.8MB download, the packages are overwriting existing files.

It’s important to understand that a basic `apt upgrade` never removes software or adds new software. As a result, it is safe to use this command to keep your system fully patched because it should never break things. However, occasionally you will see the “0 not upgraded” status change, which means some things cannot be upgraded. This happens when some software must be installed or removed to satisfy the dependencies of the updated package, which, as previously mentioned, an `apt upgrade` never does. In this situation, you need to use `apt dist-upgrade`, so named because it’s designed to allow users to upgrade from one version of Debian/Ubuntu to a newer version—an upgrade that inevitably involves changing just about everything on the system, removing obsolete software, and installing the latest features. This is one of the most-loved features of Debian because it enables you to move from version to version without your having to download and install new CDs. Keeping regular upgrades and distro upgrades separate is very useful for making sure that security updates and simple bug fixes don’t change software configurations that you may be counting on, especially on a machine that needs to be consistently available and working, such as a server.

Whereas `apt upgrade` and `apt dist-upgrade` are available for upgrading packages, `apt install` is responsible for adding new software. For example, if you want to install the MySQL database server, you run this:

```
matthew@seymour:~$ sudo apt install mysql-server
```

Internally, APT queries `mysql-server` against its list of software and finds that it matches the `mysql-server-5.7` package. It then finds which dependencies it needs that you don't already have installed and gives you a report like this one:

```
matthew@seymour:~$ sudo apt install mysql-server
[sudo] password for matthew:

Reading package lists... Done
Building dependency tree
Reading state information... Done
The following extra packages will be installed:
  libaio1 libevent-core-2.1.6 libhtml-template-perl mysql-client-5.7
  mysql-client-core-5.7 mysql-common mysql-server mysql-server-5.7
  mysql-server-core-5.7
Suggested packages:
  libipc-sharedcache-perl tinyca mailx
The following NEW packages will be installed:
  libaio1 libevent-core-2.1.6 libhtml-template-perl mysql-client-5.7
  mysql-client-core-5.7 mysql-common mysql-server mysql-server-5.7
  mysql-server-core-5.7
0 upgraded, 9 newly installed, 0 to remove and 0 not upgraded.
Need to get 20.3 MB of archives.
After this operation, 160 MB of additional disk space will be used. Do you want to
continue [Y/n]?
```

This time, you can see that APT has picked up and selected all the dependencies required to install MySQL Server 5.7, but it has also listed one recommended package and two suggested packages that it has not selected for installation. The “recommended” package is just that: The person who made the MySQL package (or its dependencies) thinks it would be a smart idea for you to also have the `mailx` package. If you want to add it, press `N` to terminate `apt` and rerun it like this:

```
matthew@seymour:~$ sudo apt install mysql-server mailx
```

The “suggested” packages are merely a lower form of recommendation. They don't add any crucial features to the software you selected for install, but it's possible that you might need them for certain noncrucial (to the main piece of software being installed) features or tasks.

NOTE

APT maintains a package cache where it stores DEB files it has downloaded and installed. This usually lives in `/var/cache/apt/archives` and can sometimes take up many hundreds of megabytes on your computer. You can have APT clean out the package cache by running `apt clean`, which deletes all the cached DEB files. Alternatively, you can run `apt autoclean`, which deletes cached DEB files that are beyond a certain age and keeps newer packages.

If you try running `apt install` with packages you already have installed, APT considers your command to be `apt update` and looks to see whether new versions are available for download.

Another important day-to-day package operation is removing things you no longer want, which you do by using the `apt remove` command, as follows:

```
matthew@seymour:~$ sudo apt remove firefox
```

Removing packages can be dangerous because APT also removes any software that relies on the packages you selected. For example, if you were to run `apt remove libgtk2.0-0` (the main graphical toolkit for Ubuntu), you would probably find that APT insists on removing more than 100 other things. The moral of the story is this: When you remove software, read the APT report carefully before pressing `y` to continue with the uninstall.

A straight `apt remove` leaves behind the configuration files of your program so that if you ever reinstall it, you do not also need to reconfigure it. If you want to remove the configuration files as well as the program files, run this command instead:

```
matthew@seymour:~$ sudo apt remove --purge firefox
```

Or run this:

```
matthew@seymour:~$ sudo apt purge firefox
```

Both of these commands will perform a full uninstall.

NOTE

You can see a more extensive list of `apt` parameters by running `apt` without any parameters. The cryptic line at the bottom, “This APT has Super Cow Powers,” is made even more cryptic if you run the command `apt moo`.

Finding Software

With so many packages available, it can be hard to find exactly what you need by using APT at the command line. The general search tool `apt-cache` is used like this:

```
matthew@seymour:~$ apt-cache search kde
```

Depending on which repositories you have enabled, the tool returns about 1,000 packages. Many of those results do not even have KDE in the package name but are matched because the description contains the word *KDE*.

You can filter through this information in several ways. First, you can instruct `apt-cache` to search only in the package names, not in their descriptions by using the `-n` parameter, like this:

```
matthew@seymour:~$ apt-cache -n search kde
```

Now the search goes down from more than 1,000 packages to a few hundred.

Another way to limit search results is to use some basic regular expressions, such as `^`, meaning “start,” and `$`, meaning “end.” For example, you might want to search for programs that are part of the main KDE suite and not libraries (usually named something like `libkde`), additional bits (such as `xmms-kde`), and things that actually have nothing to do with KDE yet still match the search (for example, `tkdesk`). Do this by searching for packages that have a name starting with `kde`, as follows:

```
matthew@seymour:~$ apt-cache -n search ^kde
```

Perhaps the easiest way to find packages is to combine `apt-cache` with `grep` to search within search results. For example, if you want to find all games-related packages for KDE, you can run this search:

```
matthew@seymour:~$ apt-cache search games | grep kde
```

When you’ve found the package you want to install, run it through `apt install` as usual. If you first want a little more information about that package, you can use `apt-cache showpkg`, like this:

```
matthew@seymour:~$ apt-cache showpkg mysql-server-5.7
```

This shows information on “reverse depends” (which packages require, recommend, or suggest `mysql-server-5.7`), “dependencies” (which packages are required, recommended, or suggested to install `mysql-server-5.7`), and “provides” (which functions this package gives you). The “provides” list is quite powerful because it allows different packages to provide given resources. For example, a MySQL database-based program requires MySQL to be installed, but it isn’t fussy about whether you install MySQL 4.1 or MySQL 5.7. In this situation, the Debian packages for MySQL 4.1 and MySQL 5.7 would both have `mysql-server-4.1` in the provides list, meaning that they would offer the functionality provided by MySQL 4.1. Therefore, you could install either version to satisfy the MySQL-based application.

Using `apt-get` Instead of `apt`

While the new, simplified interface to APT that removes the hyphen and the second part of the command works great and includes lovely updates like a progress bar, it does not yet include all functionality of the venerable `apt-get`. Table 9-1 lists some of the new commands and what they replace, in case you are more familiar with the old commands. Both versions work, so no relearning is necessary if you do not want to switch. Today, `apt-get` is used only for much more advanced package management tasks beyond the needs of what we cover in this book.

Table 9.1 apt-get Versus apt

apt-get Command	apt Command
apt-get install	apt install
apt-get remove	apt remove
apt-get update	apt update
apt-get upgrade	apt upgrade
apt-get dist-upgrade	apt full-upgrade
apt-get remove --purge	apt purge
apt-get autoremove	apt autoremove
apt-get search	apt search
apt-get show	apt show
dpkg --get-selections	apt list --installed
apt-get purge	apt purge

Compiling Software from Source

Compiling applications from source is not difficult. There are two ways to do it: You can use the source code available in the Ubuntu repositories, or you can use source code provided by upstream developers (which is most useful for projects that are not available in the Ubuntu repositories). For either method, you need to install the `build-essential` package to ensure that you have the tools you need for compilation. You may also need to install `automake` and `checkinstall`, which are build tools.

Compiling from a Tarball

Most source code that is not in the Ubuntu repositories is available from the original writer or from a company's website as compressed source *tarballs*—that is, `tar` files that have been compressed using `gzip` or `bzip`. The compressed files typically uncompress into a directory containing several files. It is always a good idea to compile source code as a regular user to limit any damage that broken or malicious code might inflict, so create a directory named `source` in your home directory.

From wherever you downloaded the source tarball, uncompress it into the `~/source` directory by using the `-C` option to `tar`:

```
matthew@seymour:~$ tar zxvf packagename.tgz -C ~/source
matthew@seymour:~$ tar zxvf packagename.tar.gz -C ~/source
matthew@seymour:~$ tar jxvf packagename.bz -C ~/source
matthew@seymour:~$ tar jxvf packagename.tar.bz2 -C ~/source
```

If you are not certain what file compression method was used, use the `file` command to figure it out:

```
matthew@seymour:~$ file packagename
```

Now, change directories to `~/source/packageName` and look for a file named `README`, `INSTALL`, or something similar. Print out the file, if necessary, because it contains specific instructions on how to compile and install the software. Typically, the procedure to compile source code is as follows:

```
matthew@seymour:~/source/packageName$ ./configure
```

This runs a script to check whether all dependencies are met and whether the build environment is correct. If you are missing dependencies, the `configure` script normally tells you exactly which ones it needs. If you have the Universe and Multiverse repositories enabled in Synaptic, chances are you will find the missing software (usually libraries) in there.

When your `configure` script succeeds, run the following to compile the software:

```
matthew@seymour:~/source/packageName$ make
```

Finally, run the following:

```
matthew@seymour:~/source/packageName$ sudo make install
```

If the compile fails, check the error messages for the reason and run the following before you start again:

```
matthew@seymour:~/source/packageName$ make clean
```

You can also run the following to remove the software if you do not like it:

```
matthew@seymour:~/source/packageName$ sudo make uninstall
```

Compiling from Source from the Ubuntu Repositories

You might sometimes want to recompile a package, even though a binary package is available in the Ubuntu repositories. For example, a program might have been compiled into a binary with a specific feature disabled that you would like to use. Here is how you can do this, using a software package named `foo` that you want to compile.

First, get the source from the Ubuntu repositories:

```
matthew@seymour:~$ apt source foo
```

Install the build dependencies for the package:

```
matthew@seymour:~$ sudo apt build-dep foo
```

Change to the directory for the source code (which may include the version number):

```
matthew@seymour:~$ cd foo-4.5.2
```

Make whatever changes you want to make to the package or to the compilation flags. You can do this by using `./configure` and `make`, or sometimes you can make manual changes

to a configuration file. Each package has the potential to do this differently, so you need to see that program's documentation. Try looking for a README file in the source code to get started.

Next, create a new `debian/changelog` entry. After you enter this command, you need to enter a message that tells why a new version was made, perhaps something like *Matthew's flight of fancy with extra sauce*.

```
matthew@seymour:~$ dch -i
```

NOTE

Ubuntu package numbering follows a specific pattern. To help yourself later, you should stick to this pattern. Using the `foo` numbers shown here, a typical Ubuntu package that was inherited from Debian with no changes would then be `4.5.2-1`. A package inherited from Debian but changed for Ubuntu would be `4.5.2-1ubuntu1` (and then `ubuntu2` for a second version, and so on). A package that did not have a version in Debian but which was created for Ubuntu would be `4.5.2-0ubuntu1` (and `ubuntu2` and so on).

Build the source package. This creates all the files necessary for uploading a package:

```
matthew@seymour:~$ debuild -S
```

Finally, you are left with a `foo-4.5.2-1ubuntu1custom.deb` package (using whatever version number or suffix you created earlier) that you can install—and later uninstall as well—using your package manager. In some instances, multiple DEB files might be created, in which case you replace the individual package name in the example here with `*.deb`:

```
matthew@seymour:~$ sudo dpkg -Oi foo-4.5.2-1ubuntu1custom.deb
```

Configuration Management

This section provides a quick introduction to a couple tools that might be useful to you if you want more control over system configuration management. For larger needs, see Chapter 33, “Managing Sets of Servers.”

dotdee

If you run Linux-based systems, you will find a series of directories that end with a `.d` and that store configuration files. These are sometimes called `.d` or “dotdee” directories. If you look in `/etc/`, you find many such directories (such as `apparmor.d` and `pam.d`). Opening these directories reveals a large number of configuration files and perhaps other directories containing even more. In Ubuntu or other Debian-based systems, it is a violation of etiquette (and Debian policy) for any software package to be allowed to directly change the configuration files of another package. This can be problematic if you want to use system configuration management software.

dotdee solves this problem by allowing you to replace any flat file in your file system with a symlink pointing to a file that is generated from a `.d`-style directory. It saves the original file and then updates the generated file automatically and dynamically any time any file in the original `.d` directory is added, deleted, or modified. This way, the Debian policy and general etiquette standards are met, but configurations can be modified as needed by an external program.

dotdee works its magic using `inotify` to dynamically and instantly update the master file. The master file can be built three different ways: using flat files, which are concatenated; using diff/patch files, which are applied in a quiltlike manner; and using executables, which process `stdin` and dump to `stdout`. This flexibility should make any system administrator or developer guru happy.

Ubuntu Core

Ubuntu Core adds the absolute minimum of files and code necessary for a usable Ubuntu server image to it as a new means of managing software packages. The idea is similar to how smart phones like Android-based phones provide software: The software packages include everything they need to run on the operating system, effectively making it so that a package is isolated from the operating system more completely. This is designed to protect against the possibility of a package breaking other packages or an entire operating system installation. It is also intended to make updates easier and cleaner for use cases like cloud deployment or IoT (Internet of things). This feature first appeared in Ubuntu 16.04 LTS.

Using the Snappy Package Manager

Software bundles that can be packaged using the Snappy package manager are called *snaps*. Snaps can be installed using Ubuntu Software or from the command line, as described in this section.

To show a list of snap packages that are available to be installed, use the following:

```
matthew@seymour:~$ snap find
```

Because snaps are new, few packages are available today. However, the list is growing and is likely to become unwieldy at some point. Sure, you can use `grep` to search through the output to try to find a specific package in the list, but you can also use this:

```
matthew@seymour:~$ snap find searchterm
```

To install a snap package, use the following:

```
matthew@seymour:~$ sudo snap install packagename
```

To show a list of snap packages that are currently installed, use the following:

```
matthew@seymour:~$ snap list
```

To update a snap package, use the following:

```
matthew@seymour:~$ sudo snap refresh packagename
```

To remove a snap package, use the following:

```
matthew@seymour:~$ sudo snap remove packagename
```

To display a list of changes, such as when snaps were installed, updated, or removed, use the following:

```
matthew@seymour:~$ snap changes
```

References

- ▶ www.debian.org/doc/manuals/project-history/ch-detailed.en.html—History of the Debian Linux package system
- ▶ www.nongnu.org/synaptic/—Home of the Synaptic package manager

CHAPTER 10

Command-Line Beginner's Class

The Linux command line is one of the most powerful tools available for computer system administration and maintenance. The command line is also known as the terminal, shell, console, command prompt, and *command-line interface (CLI)*. For the purposes of this chapter and the next, these terms are interchangeable, although fine-grained differences do exist between them.

Using the command line is an efficient way to perform complex tasks accurately and much more easily than it would seem at a first glance. Knowledge of the commands available to you and also how to string them together makes using Ubuntu easier for many tasks. Many of the commands were created by the GNU Project as free software analogs to previously existing proprietary UNIX commands. You can learn more about the GNU Project at www.gnu.org/gnu/thegnuproject.html.

This chapter covers some of the basic commands that you need to know to be productive at the command line. You find out how to get to the command line and discover some of the commands used to navigate the file system and perform basic operations with files, directories, and users. This chapter does not give comprehensive coverage of all the commands discussed, but it does give you enough to get started. Chapter 11, “Command-Line Master Class, Part 1,” advances the subject further and expands on some of the commands from this chapter. The skills you discover in this chapter help you get started using the command line with confidence.

IN THIS CHAPTER

- ▶ What Is the Command Line?
- ▶ Accessing the Command Line
- ▶ User Accounts
- ▶ Reading Documentation
- ▶ Understanding the Linux File System Hierarchy
- ▶ Navigating the Linux File System
- ▶ Working with Permissions
- ▶ Working with Files
- ▶ Working as Root
- ▶ Commonly Used Commands and Programs
- ▶ References

What Is the Command Line?

If you have spent any amount of time with experienced Linux users, you have heard them mention the command line. Some, especially those who began their journey in the Linux world using distributions that make it easy to complete many tasks using a *graphical user interface (GUI)*, such as Ubuntu, might speak with trepidation about the mysteries of the text interface. Others either praise its power or comment about doing something via the command line as if it were the most natural and obvious way to complete a task.

Embracing either extreme is not necessary. You might develop an affinity for the command line when performing some tasks and might prefer using the GUI for others. This is where most users end up today. Some might say that you will never need to access the command line because Ubuntu offers a slew of graphical tools that enable you to configure most things on your system. Although the premise might be true most of the time, some good reasons exist to acquire a fundamental level of comfort with the command line that you should consider before embracing that view.

Sometimes things go wrong, and you might not have the luxury of a graphical interface to work with. In such situations, a fundamental understanding of the command line and its uses can be a real lifesaver. Also, some tasks end up being far easier and faster to accomplish from the command line. More importantly, though, you will be able to make your way around a command-line-based system, which you will encounter if you ever work with a Linux server because most Linux servers have no GUI, and all administration is done using a command-line interface.

NOTE

Don't be tempted to skip over this chapter as irrelevant. You should take the time to work through the chapter and ensure that you are comfortable with the command line before moving on. Doing so will benefit you greatly for years to come.

Initially, you might be tempted to think of the command line as the product of some sort of black and arcane art; in some ways, it can appear to be extremely difficult and complicated to use. However, with a little perseverance, by the end of this chapter, you will start to feel comfortable using the command line, and you'll be ready to move on to Chapter 11, "Command-Line Master Class, Part 1," and Chapter 12, "Command-Line Master Class, Part 2."

This chapter introduces you to commands that enable you to perform the following:

- ▶ **Routine tasks**—Logging in and out, changing passwords, and listing and navigating file directories
- ▶ **Basic file management**—Creating files and folders, copying or moving them around the file system, and renaming and deleting them
- ▶ **Basic system management**—Shutting down or rebooting, changing file permissions, and reading man pages, which are entries for commands included as files already on your computer in a standardized manual format

The information in this chapter is valuable for individual users and system administrators who are new to Linux and are learning to use the command line for the first time.

TIP

If you have used a computer for many years, you have probably come into contact with MS-DOS, in which case being presented with a black screen will fill you with a sense of nostalgia. Don't get too comfy; the command line in Linux is different from (and actually more powerful than) its distant MS-DOS cousin. Even cooler is that whereas MS-DOS skills are transferable only to other MS-DOS environments, the skills you learn at the Linux command line can be transferred easily to other UNIX and UNIX-like operating systems, such as Solaris, OpenBSD, FreeBSD, and even macOS, which provides access to the terminal.

Accessing the Command Line

You can quickly access the terminal by using the desktop menu option Terminal. This opens `gnome-terminal`, from which you can access the terminal while remaining in a GUI environment. This time, the terminal appears as white text on an aubergine (dark purple) background. This is the most common method for accessing the terminal for most desktop users.

NOTE

Finding and running programs, such as Terminal, from a GUI is covered in Chapter 4, “Ubuntu Desktop Options,” as is logging in to a Linux system using a graphical interface. This chapter focuses on text-based logins and the use of Linux.

Text-Based Console Login

If you access the terminal as described previously, by opening the Terminal app, you will not need to log in because you are already logged in to your account.

If you are accessing the terminal remotely, such as from another computer using SSH (see Chapter 19, “Remote Access with SSH and VNC”) you will start with a prompt similar to this one:

```
login:
```

Your prompt might vary, depending on the version of Ubuntu you are using and the method you are using to connect. In any event, at this prompt, type in your username and press Enter. When you are prompted for your password, type it in and press Enter.

NOTE

Your password is not echoed back to you, which is a good idea. Why is it a good idea? Security. This prevents any shoulder surfers from seeing what you've typed or the length of the password.

Pressing the Enter key drops you to a shell prompt, signified by the dollar sign:

```
matthew@seymour:~$
```

This particular prompt says that the user is logged in as the user `matthew` on the system `seymour` and is currently in the home directory; Linux uses the tilde (`~`) as shorthand for the home directory, which would usually be something like `/home/matthew`.

Note that throughout this book we use the default shell for the command line, but other shells exist. The default and other shells are discussed in more detail in Chapter 14, “Automating Tasks and Shell Scripting.”

TIP

Navigating through the system at the command line can get confusing at times, especially when a directory name occurs in several places. Fortunately, Linux includes a simple command that tells you exactly where you are in the file system. It's easy to remember that this command is `pwd` because it is an abbreviation of *print working directory*. You simply type `pwd` at any point to get the full path of your location. For example, typing `pwd` after following the preceding instructions shows `/home/yourusername`, meaning that you are currently in your home directory.

Using the `pwd` command can save you a lot of frustration when you have changed directory half a dozen times and have lost track of where you are.

Logging Out

Use the `exit` or `logout` command or press `Ctrl+D` to exit your session. You are then returned to the login prompt. If you use virtual consoles, remember to exit each console before leaving your PC. (Otherwise, someone could easily sit down and use your account.)

Logging In and Out from a Remote Computer

You can happily log in on your computer, an act known as a *local* login. In addition, you can log in to your computer via a network connection from a remote computer. Linux-based operating systems provide a number of remote access commands you can use to log in to other computers on your *local area network (LAN)*, *wide area network (WAN)*, or the Internet. Note that you must have an account on the remote computer, *and* the remote computer must be configured to support remote logins; otherwise, you won't be able to log in.

NOTE

See Chapter 18, “Networking,” to see how to set up network interfaces with Linux to support remote network logins and Chapter 19 to see how to start remote access services (such as `ssh`).

The best and most secure way to log in to a remote Linux computer is to use `ssh`, the Secure Shell client. Your login and session are then encrypted while you work on the

remote computer. The `ssh` client features many command-line options but can be simply used with the name or IP address of the remote computer, as follows:

```
matthew@seymour:~$ ssh 192.168.0.41
The authenticity of host '192.168.0.41 (192.168.0.41)' can't be established.
RSA key fingerprint is e1:db:6c:da:3f:fc:56:1b:52:f9:94:e0:d1:1d:31:50.
Are you sure you want to continue connecting (yes/no)?
```

yes

The first time you connect with a remote computer using `ssh`, Linux displays the remote computer's encrypted identity key and asks you to verify the connection. After you type **yes** and press Enter, you are warned that the remote computer's identity (key) has been entered in a file named `known_hosts` under the `.ssh` directory in your home directory. You are also prompted to enter your password:

```
Warning: Permanently added '192.168.0.41' (RSA) \
to the list of known hosts.
matthew@192.168.0.41's password:
matthew@babbage~$
```

After entering your password, you can work on the remote computer, which you can confirm by noticing the changed prompt that now uses the name of the remote computer on which you are working. Again, because you are using `ssh`, everything you enter on the keyboard in communication with the remote computer is encrypted. When you log out, you return to the shell on your computer:

```
matthew@babbage~$ logout
matthew@seymour:~$
```

User Accounts

A good place to start this section is with the concept of user-based security. For the most part, only two types of people access the system as users. (Although there are other accounts that run programs and processes, here we are talking about accounts that represent human beings rather than something like an account created for a web server process.) Most people have a regular user account. These users can change anything that is specific to their accounts, such as the wallpaper on the desktop, their personal preferences, and the configuration for a program when it is run by them using their account. Note that the emphasis is on anything that is *specific to their accounts*. This type of user cannot make system-wide changes that could affect other users.

To make system-wide changes, you need to use super user privileges, such as can be done using the account you created when you started Ubuntu for the first time (see Chapter 1, “Installing Ubuntu and Post-Installation Configuration”). With super user privileges you have access to the entire system and can carry out any task—even destructive tasks. To help prevent this from happening, this user does not run with these powers enabled at all times but instead spends most of the time as a regular user.

To use super user privileges from the command line, you need to preface the command you want to execute with another command, `sudo`, followed by a space and the command you want to run. As a mnemonic device, some think of this as “super user do.” When you press Enter (after typing the remaining command), you are prompted for your password, which you should type, and then press the Enter key. As usual on any UNIX-based system, the password does not appear on the screen while you are typing it as a security measure, in case someone is watching over your shoulder. Ubuntu then carries out the command but with super user privileges.

An example of the destructive nature of working as the super user is the age-old example `sudo rm -rf /`, which would erase everything on your hard drive if it did not require appending `--no-preserve-root` to work—which was added specifically to prevent people from accidentally doing this. If you enter a command using `sudo` as a regular user who does not have an account with super user privileges, an error message appears, and nothing happens because the command will not run. We recommend that you don't try this particular command as a test, though. If you enter this command using an account with super user privileges, you will soon find yourself starting over with a fresh installation and hoping you have a current backup of all your data. You need to be especially careful when using your super user privileges; otherwise, you might do irreparable damage to your system.

However, the ability to work as the super user is fundamental to a healthy Linux system and should not be feared but rather respected, even while used only with focused attention. Without this ability, you could not install new software, edit system configuration files, or do a large number of important administration tasks. By the way, you have already been performing operations with super user privileges from the GUI if you have ever been asked to enter your password to complete a specific task, such as installing software updates. The difference is that most graphical interfaces limit the options that users have and make it a little more difficult to do some of the big, disruptive tasks, even the ones that are incredibly useful.

Ubuntu works slightly differently from many other Linux distributions. If you study some other Linux distros, especially older or more traditional ones, you will hear about a specific user account called `root`, which is a super user account. In those distros, instead of typing `sudo` before a command while using a regular user account with super user privileges, you log in to the `root` account and issue the command without entering a password (at least by default; in almost all cases, `sudo` can be installed and configured in these distros). In those cases, you can tell when you are using the `root` account at the command line because you see a pound sign (`#`) in the command-line prompt in place of the dollar sign (`$`). For example, you see `matthew@seymour:~#` instead of the usual `matthew@seymour:~$` prompt.

In Ubuntu, the `root` account is disabled by default because forcing regular users with super user privileges to type a specific command every time they want to execute a command as a super user should have the benefit of making them carefully consider what they are doing when they use that power. It is easy to forget to log out of a `root` account,

and entering a powerful command while logged in to root can be catastrophic. However, if you are more experienced and comfortable with the more traditional method of using super user privileges and want to enable the root account, you can use the command `sudo passwd`. When prompted, enter your user password to confirm that your user account has super user privileges. You are then asked for a new UNIX password, which will be the password for the root account, so make sure to remember it. You are also prompted to repeat the password, in case you've made any mistakes. After you've typed it in and pressed Enter, the root account is active. You find out how to switch to root later on.

An alternative way of getting a root prompt, without having to enable the root account, is to issue the command `sudo -i`. After entering your password, you find yourself at a root prompt (`#`). Do what you need to do, and when you are finished, type `exit` and press Enter to return to your usual prompt. You can learn more about `sudo` and root from an Ubuntu perspective at <https://help.ubuntu.com/community/RootSudo>.

Reading Documentation

Although you learn the basics of using Ubuntu in this book, you need time and practice to master and troubleshoot more complex aspects of the Linux operating system and your distribution. As with any other operating system, you can expect to encounter some problems or perplexing questions as you continue to work with Linux. The first place to turn for help with these issues is the documentation included with your system; if you cannot find the information you need there, check Ubuntu's website.

Using Man Pages

To learn more about a command or program, use the `man` command followed by the name of the command. Man pages are stored in places like `/usr/share/man` and `/usr/local/share/man`, but you don't need to know that. To read a man page, such as the one for the `rm` command, use the `man` command like this:

```
matthew@seymour:~$ man rm
```

After you press Enter, the `less` command (a Linux command known as a *pager*) displays the man page. The `less` command is a text browser you can use to scroll forward and backward (even sideways) through the document to learn more about the command. Type the letter `h` to get help, use the forward slash (`/`) to enter a search string, or press `q` to quit.

No one can remember everything. Even the best and most experienced systems administrators use man pages regularly. Looking up complicated information is easy because this frees you from having to recall it all, enabling you to focus on your task rather than punishing you for not remembering syntax.

NOTE

Nearly every one of the hundreds of commands included with Linux has a man page; however, some do not have man pages or have only simple ones. You can use the `info` command to read more detailed information about some commands or as a replacement for

others. For example, to learn even more about `info` (which has a rather extensive manual page), use the `info` command like this:

```
matthew@seymour:~$ info info
```

Use the arrow keys to navigate through the document and press `q` to quit reading.

Using apropos

Linux, like UNIX, is a self-documenting system, with man pages accessible through the `man` command. Linux offers many other helpful commands for accessing its documentation. You can use the `apropos` command (for example, with a keyword such as `partition`) to find commands related to partitioning, like this:

```
matthew@seymour:~$ apropos partition
addpart      (8)  - Simple wrapper around the "add partition" ioctl
all-swaps    (7)  - Event signaling that all swap partitions have been ac...
cfdisk       (8)  - Curses/slang based disk partition table manipulator fo...
delpart      (8)  - Simple wrapper around the "del partition" ioctl
fdisk        (8)  - Partition table manipulator for Linux
gparted      (8)  - Gnome partition editor for manipulating disk partitions.
Mpartition   (1)  - Partition an MSDOS hard disk
Partprobe    (8)  - Inform the OS of partition table changes
Partx        (8)  - Telling the kernel about presence and numbering of on...
Pvcreate     (8)  - Initialize a disk or partition for use by LVM
Pvresize     (8)  - Resize a disk or partition in use by LVM2
Sfdisk       (8)  - Partition table manipulator for Linux
```

For the record, `apropos` is the equivalent of using `man -k`.

Using whereis

To find a command and its documentation, you can use the `whereis` command. For example, if you are looking for the `fdisk` command, you can use this:

```
matthew@seymour:~$ whereis fdisk
fdisk: /sbin/fdisk /usr/share/man/man8/fdisk.8.gz
```

Understanding the Linux File System Hierarchy

Linux has inherited from UNIX a well-planned hierarchy for organizing things. It isn't perfect, but it is generally logical and mostly consistent, although distributions do tend to make some modifications that force some thinking and adaptation when moving between, say, Fedora, Slackware, and Ubuntu. Table 10.1 shows some of the top-level directories that are part of a standard Linux distro.

Table 10.1 Basic Linux Directories

Directory	Description
/	The root directory
/bin	Essential commands
/boot	Boot loader files, Linux kernel
/dev	Device files
/etc	System configuration files
/home	User home directories
/lib	Shared libraries, kernel modules
/lost+found	Recovered files (if found after a file system check)
/media	Mount point for removable media, such as DVDs and floppy disks
/mnt	Usual mount point for local, remote file systems, file systems that are additional to the standard, such as a DVD-ROM or another HDD
/opt	Add-on software packages
/proc	Kernel information, process control
/root	Super user (root) home
/sbin	System commands (mostly root only)
/sys	Real-time information on devices used by the kernel
/tmp	Temporary files
/usr	Software not essential for system operation, such as applications
/var	Variable files relating to services that run on the system, but whose contents are expected to change regularly during normal operation

Knowing these directories can help you find files when you need them. This knowledge can even help you partition hard drives when you install new systems; you can choose to put certain directories on their own distinct partition, which can be useful for things like isolating directories from one another (for example, for server security you might put a directory like `/boot` that doesn't change often on its own partition and make it read-only and unchangeable without specific operations being done by a super user during a maintenance cycle). Desktop users probably don't need to think about such operations, but knowing the directory tree is still quite useful when you want to find the configuration file for a specific program and set some program options system-wide to affect all users.

NOTE

This is a lot to remember, especially at first. For reference, there is a man page for the Linux file system hierarchy:

```
matthew@seymour:~$ man hiercssn
```

This returns a detailed listing, with a description of each part.

More information about file systems is available in Chapter 21, "Performance Tuning."

Some of the important directories in Table 10.1, such as those containing user and root commands or system configuration files, are discussed in the following sections. You may use and edit files under these directories when you use Ubuntu.

Essential Commands in `/bin` and `/sbin`

The `/bin` directory contains essential commands used by the system for running and booting the system. In general, only the root operator uses the commands in the `/sbin` directory. The software in both locations is essential to the system; it makes the system what it is, and changing or removing this software could cause instability or a complete system failure. Often, the commands in these two directories are *statically* linked, which means the commands do not depend on software libraries residing under the `/lib` or `/usr/lib` directories. Nearly all the other applications on your system are *dynamically* linked, meaning that they require the use of external software libraries (also known as *shared* libraries) to run. This is a feature for both sets of software.

The commands in `/bin` and `/sbin` are kept stable to maintain foundational system integrity and do not need to be updated often, if at all. For the security of the system, these commands are kept in a separate location and isolated so that changes are more difficult and so it will be more obvious to the system administrator if unauthorized changes are attempted or made.

Application software changes more frequently, and applications often use the same functions that other pieces of application software use. This was the genesis of shared libraries. When a security update is needed for something that is used by more than one program, it has to be updated in only one location, a specific software library. This enables easy and quick security updates that will affect several pieces of non-system-essential software at the same time by updating one shared library, contained in one file on the computer.

Configuration Files in `/etc`

System configuration files and directories reside under the `/etc` directory. Some major software packages, such as Apache, OpenSSH, and `xinetd`, have their own subdirectories in `/etc`, filled with configuration files. Others, like `crontab` or `fstab`, use one file. Examples of system-related configuration files in `/etc` include the following:

- ▶ **fstab**—The file system table is a text file that lists each hard drive, CD-ROM, or other storage device attached to your PC. The table indexes each device's partition information with a place in your Linux file system (directory layout) and lists other options for each device when used with Linux (see Chapter 22, "Kernel and Module Management"). Nearly all entries in `fstab` can be manipulated by root using the `mount` command.
- ▶ **modprobe.d/**—This folder holds all the instructions to load kernel modules that are required as part of system startup.
- ▶ **passwd**—This file holds the list of users for the system, including special-purpose nonhuman users like `syslog` and `CouchDB`, along with user account information.
- ▶ **sudoers**—This file holds a list of users or user groups with super user access.

User Directories: /home

The most important data on a non-server Linux system often resides in the user's directories, found under the `/home` directory. User directories are named by default according to account usernames, so on a computer where you have an account named `matthew`, your home directory would generally be found in `/home/matthew`. This can be changed, as described in Chapter 11.

Segregating the system and user data can be helpful in preventing data loss and making the process of backing up easier. For example, having user data reside on a separate file system or mounted from a remote computer on the network might help shield users from data loss in the event of a system hardware failure. For a laptop or desktop computer at home, you might place `/home` on a separate partition from the rest of the file system so that if the operating system is upgraded, damaged, or reinstalled, `/home` would be more likely to survive the event intact.

Using the Contents of the /proc Directory to Interact with or Obtain Information from the Kernel

The contents of the `/proc` directory are created from memory and exist only while Linux is running. This directory contains special files that either extract information from or send information to the kernel. Many Linux utilities extract information from dynamically created directories and files under this directory, also known as a *virtual file system*. For example, the `free` command obtains its information from a file named `meminfo`:

```
matthew@seymour:~$ free
```

	total	used	free	shared	buffers	cached
Mem:	4055680	2725684	1329996	0	188996	1551464
-/+ buffers/cache:		985224	3070456			
Swap:	8787512	0	8787512			

This information constantly changes as the system is used. You can get the same information by using the `cat` command to see the contents of the `meminfo` file:

```
matthew@seymour:~$ cat /proc/meminfo
MemTotal:                4055680 KB
MemFree:                  1329692 KB
Buffers:                  189208 KB
Cached:                   1551488 KB
SwapCached:               0 KB
Active:                   1222172 KB
Inactive:                 1192244 KB
Active(anon):              684092 KB
Inactive(anon):            16 KB
Active(file):              538080 KB
Inactive(file):           1192228 KB
Unevictable:              48 KB
```

```

Mlocked:                48 KB
SwapTotal:              8787512 KB
SwapFree:               8787512 KB
Dirty:                  136 KB
Writeback:              0 KB
AnonPages:              673760 KB
Mapped:                 202308 KB
Shmem:                  10396 KB
Slab:                   129248 KB
SReclaimable:          107356 KB
SUnreclaim:             21892 KB
KernelStack:           2592 KB
PageTables:             30108 KB
NFS_Unstable:           0 KB
Bounce:                 0 KB
WritebackTmp:           0 KB
CommitLimit:            10815352 KB
Committed_AS:           1553172 KB
VmallocTotal:           34359738367 KB
VmallocUsed:             342300 KB
VmallocChunk:           34359387644 KB
HardwareCorrupted:      0 KB
HugePages_Total:        0
HugePages_Free:         0
HugePages_Rsvd:         0
HugePages_Surp:         0
Hugepagesize:           2048 KB
DirectMap4k:            38912 KB
DirectMap2M:            4153344 KB

```

The `/proc` directory can also be used to dynamically alter the behavior of a running Linux kernel by “echoing” numerical values to specific files under the `/proc/sys` directory. For example, to “turn on” kernel protection against one type of denial-of-service (DoS) attack known as *SYN flooding*, use the `echo` command to send the number 1 to the following `/proc` path:

```
matthew@seymour:~$ sudo echo 1 >/proc/sys/net/ipv4/tcp_syncookies
```

Other ways to use the `/proc` directory include the following:

- ▶ Getting CPU information, such as the family, type, and speed from `/proc/cpuinfo`.
- ▶ Viewing important networking information under `/proc/net`, such as active interfaces information in `/proc/net/dev`, routing information in `/proc/net/route`, and network statistics in `/proc/net/netstat`.

- ▶ Retrieving file system information.
- ▶ Reporting media mount point information via USB; for example, the Linux kernel reports what device to use to access files (such as `/dev/sda`) if a USB camera or hard drive is detected on the system. You can use the `dmesg` command to see this information.
- ▶ Getting the kernel version in `/proc/version`, performance information such as uptime in `/proc/uptime`, or other statistics such as CPU load, swap file usage, and processes in `/proc/stat`.

Working with Shared Data in the `/usr` Directory

The `/usr` directory contains software applications, libraries, and other types of shared data for use by anyone on the system. Many Linux system administrators give `/usr` its own partition. A number of subdirectories under `/usr` contain manual pages (`/usr/share/man`), software package shared files (`/usr/share/name_of_package`, such as `/usr/share/emacs`), additional application or software package documentation (`/usr/share/doc`), and an entire subdirectory tree of locally built and installed software, `/usr/local`.

Temporary File Storage in the `/tmp` Directory

As its name implies, the `/tmp` directory is used for temporary file storage. As you use Linux, various programs create files in this directory. Files in this directory are cleared daily by a `cron` job and every time the system is booted.

Accessing Variable Data Files in the `/var` Directory

The `/var` directory contains subdirectories used by various system services for spooling and logging. Many of these variable data files, such as print spooler queues, are temporary, whereas others, such as system and kernel logs, are renamed and rotated in use. Incoming email is usually directed to files under `/var/spool/mail`.

NOTE

Linux also uses `/var` for other important system services, such as the Apache web server's initial home page directory for the system, `/var/www/html`. (See Chapter 25, "Apache Web Server Management," for more information about using Apache.)

Navigating the Linux File System

In the Linux file system, as with its predecessor UNIX, everything is a file: data files, binary files, executable programs, and even input and output devices. These files are placed in a series of directories that act like file folders. A directory is nothing more than a special type of file that contains a list of other files/directories. These files and directories

are used to create a hierarchical structure that enables logical placement of specific types of files. Later this chapter discusses the standard hierarchy of the Linux file system. First, you learn how to navigate and interact with the file system.

NOTE

A directory with contents is called a *parent*, and its contents are called *children*, as in “/home/matthew/Documents is a child directory of /home/matthew, its parent.”

Listing the Contents of a Directory with `ls`

The `ls` command lists the contents of the current directory. It is commonly used by itself, but a number of options (also known as switches) are available for `ls` and give you more information. If you have just logged in as described earlier, the `ls` command lists the files and directories in your home directory:

```
matthew@seymour:~$ ls
Documents  Music      file.txt  Pictures  Music
```

NOTE

All directory listings in this chapter are abbreviated to save space.

By itself, the `ls` command shows just a list of names. Some are files, some are directories. This is useful if you know what you are looking for but cannot remember the exact name. However, using `ls` in this manner has some limitations. First, it does not show hidden files. Hidden files use filenames that start with a period (.) as the first character. They are often used for configuration of specific programs and are not accessed frequently. For this reason, they are not included in a basic directory listing. You can see all the hidden files by adding a switch to the command, like this:

```
matthew@seymour:~$ ls -a
.          .bash_logout  Documents     Music
..         .bashrc       file.txt      Pictures
.bash_history .config       .local        .profile
```

There is still more information available about each item in a directory. To include details such as the file/directory permissions, owner and group (discussed later in this chapter), size, and the date and time it was last modified, enter the following:

```
matthew@seymour:~$ ls -al
total 608
drwxr-xr-x 38 matthew matthew 4096 2015-06-04 08:20 .
drwxr-xr-x  3 root    root   4096 2015-05-16 16:48 ..
-rw-----  1 matthew matthew  421 2015-06-04 10:27 .bash_history
```

```

-rw-r--r-- 1 matthew matthew 220 2015-05-16 16:48 .bash_logout
-rw-r--r-- 1 matthew matthew 3353 2015-05-16 16:48 .bashrc
drwxr-xr-x 13 matthew matthew 4096 2015-05-21 10:42 .config
drwxr-xr-x 2 matthew matthew 4096 2015-05-16 17:07 Documents
-rw-r--r-- 1 matthew matthew 335 2015-05-16 16:48 file.txt
drwxr-xr-x 3 matthew matthew 4096 2015-05-16 17:07 .local
drwxr-xr-x 2 matthew matthew 4096 2015-05-16 17:07 Music
drwxr-xr-x 3 matthew matthew 4096 2015-05-16 18:07 Pictures
-rw-r--r-- 1 matthew matthew 675 2015-05-16 16:48 .profile
    
```

The listing (abbreviated here) is now given with one item per line but with multiple columns. The listing starts with the number of items in the directory. (Both files and subdirectories are included; remember that the listing here is abbreviated.) Then, the details are as shown in Figure 10.1.

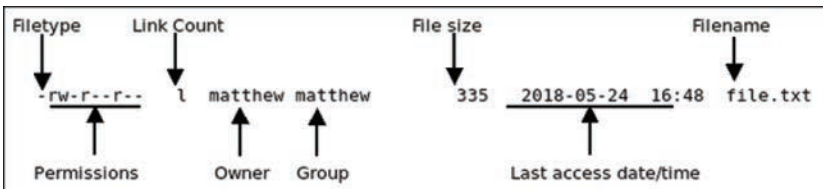


FIGURE 10.1 Decoding the output of a detailed directory listing.

These details are discussed more completely later in the chapter, in the “Working with Permissions” section.

Another useful switch is this:

```
matthew@seymour:~$ ls -R
```

This command scans and lists all the contents of the subdirectories of the current directory. This is likely to be a lot of information, so you might want to redirect the output to a text file so that you can browse through it at your leisure by using the following:

```
matthew@seymour:~$ ls -laR > listing.txt
```

TIP

The previous command sends the output of `ls -laR` to a file called `listing.txt` and demonstrates part of the power of the Linux command line. At the command line, you can use files as inputs to commands, or you can generate files as outputs, as shown. For more information about redirects and combining commands, see Chapter 14. In the meantime, note that you can read the contents of the text file by using the command `less listing.txt`, which lets you read the file bit by bit, using the arrow keys to navigate in the file (or Enter to move to the next line), the spacebar to move to the next page, and q to exit when done.

Changing Directories with `cd`

Use the `cd` command to move within the file system from one directory to another. It might help you remember this command to think of it meaning *change directory*. The most basic usage of `cd` is as follows:

```
matthew@seymour:~$ cd somedir
```

This command looks in the current directory for the `somedir` subdirectory and then moves you into it. You can also specify an exact location for a directory, like this:

```
matthew@seymour:~$ cd /home/matthew/stuff/somedir
```

You can also use the `cd` command with several shortcuts. For example, to quickly move up to the *parent* directory, the one above the one you are currently in, use the `cd` command like this:

```
matthew@seymour:~$ cd ..
```

To return to your home directory from anywhere in the Linux file system, use the `cd` command like this:

```
matthew@seymour:~$ cd
```

You can also use the `$HOME` shell environment variable to accomplish the same thing. Environment variables are discussed in greater detail in Chapter 12. To return to your home directory, type this command and press Enter:

```
matthew@seymour:~$ cd $HOME
```

You can accomplish the same thing by using the tilde (`~`), like this:

```
matthew@seymour:~$ cd ~
```

Finding Your Current Directory with `pwd`

Use `pwd` to determine where you are within the file system:

```
matthew@seymour:~$ pwd
```

Working with Permissions

Under Linux (and UNIX), everything in the file system, including directories and devices, is a file. And every file on your system has an accompanying set of permissions based on ownership. These permissions provide data security by giving specific permission settings to every single item denoting who may read, write, or execute the file. These permissions are set individually for the file's owner, for members of the group the file belongs to, and for all others on the system.

You can examine the default permissions for a file you create by using the `umask` command, which lists default permissions using the number system explained next, or by using the `touch` command and then the `ls` command's long-format listing, like this:

```
matthew@seymour:~$ touch file
matthew@seymour:~$ ls -l file
-rw-r--r-- 1 matthew matthew 0 2015-06-30 13:06 file
```

In this example, the `touch` command quickly creates a file. The `ls` command then reports on the file, displaying the following (from left to right):

- ▶ **The type of file created**—Common indicators of the type of file are in the leading letter in the output. A blank (which is represented by a dash, as in the preceding example) designates a plain file, `d` designates a directory, `c` designates a character device (such as `/dev/ttyS0`), `l` indicates a symbolic link, and `b` is used for a block device (such as `/dev/sda`).
- ▶ **Permissions**—Read, write, and execute permissions may be assigned for the owner, group, and all others on the system. (You learn more about these permissions later in this section.) Permissions are traditionally called the *mode*, which is where the `chmod` command we will discuss later comes from.
- ▶ **Number of hard links to the file**—The number `1` designates that there is only one file, and any other number indicates that there might be one or more hard-linked files. Links are created with the `ln` command. A hard-linked file is a pointer to the original file, which might be located elsewhere on the system. Only the root operator can create a hard link of a directory.
- ▶ **The owner**—This is the account that owns the file; it is originally the file creator, but you can change this designation by using the `chown` command.
- ▶ **The group**—This is the group of users allowed to access the file; it is originally the file creator's main group, but you can change this designation by using the `chgrp` command.
- ▶ **File size and creation/modification date**—The last two elements indicate the size of the file in bytes and the date the file was created or last modified.

Assigning Permissions

Under Linux, permissions are grouped by owner, group, and others, with read, write, and execute permission assigned to each, as follows:

Owner	Group	Others
<code>rwX</code>	<code>rwX</code>	<code>rxw</code>

Permissions can be indicated by mnemonic or octal characters. Mnemonic characters are listed here:

- ▶ `r` indicates permission for an owner, a member of the owner's group, or others to open and read the file.

- ▶ `w` indicates permission for an owner, a member of the owner's group, or others to open and write to the file.
- ▶ `x` indicates permission for an owner, a member of the owner's group, or others to execute the file (or read a directory).

In the previous example for the file named `file`, the owner, `matthew`, has read and write permission. Any member of the group named `matthew` may only read the file. All other users may only read the file. Also note that default permissions for files created by the root operator (while using `sudo` or a root account) will differ because of `umask` settings assigned by the shell.

Many users prefer to use numeric codes, based on octal (base 8) values, to represent permissions. Here's what these values mean:

- ▶ `4` indicates read permission.
- ▶ `2` indicates write permission.
- ▶ `1` indicates execute permission.

In octal notation, the previous example file has a permission setting of `644` (read + write or `4 + 2`, read-only or `4`, read-only or `4`). Although you can use either form of permissions notation, octal is easy to use quickly when you visualize and understand how permissions are numbered.

NOTE

In Linux, you can create groups to assign a number of users access to common directories and files, based on permissions. You might assign everyone in accounting to a group named `accounting` and allow that group access to accounts payable files while disallowing access by other departments. Defined groups are maintained by the root operator, but you can use the `newgrp` command to temporarily join other groups to access files (as long as the root operator has added you to the other groups). You can also allow or deny other groups' access to your files by modifying the group permissions of your files.

Directory Permissions

Directories are also files under Linux. For example, again use the `ls` command to show permissions, like this:

```
matthew@seymour:~$ mkdir directory
matthew@seymour:~$ ls -ld directory
drwxr-xr-x 2 matthew matthew 4096 2015-06-30 13:23 directory
```

In this example, the `mkdir` command is used to create a directory. The `ls` command, with its `-ld` option, is used to show the permissions and other information about the directory (not its contents). Here you can see that the directory has permission values of `755` (read + write + execute or `4 + 2 + 1`, read + execute or `4 + 1`, and read + execute or `4 + 1`).

This shows that the owner can read and write to the directory and, because of execute permission, also list the directory's contents. Group members and all other users can list only the directory contents. Note that directories require execute permission for anyone to be able to view their contents.

You should also notice that the `ls` command's output shows a leading `d` in the permissions field. This letter specifies that this file is a directory; normal files have a blank field in its place. Other files, such as those specifying a block or character device, have a different letter.

For example, if you examine the device file for a Linux serial port, you see the following:

```
matthew@seymour:~$ ls -l /dev/ttyS0
crw-rw---- 1 root dialout 4, 64 2015-06-30 08:13 /dev/ttyS0
```

Here, `/dev/ttyS0` is a character device (such as a serial communications port and designated by a `c`) owned by root and available to anyone in the `dialout` group. The device has permissions of `660` (read + write, read + write, no permission).

On the other hand, if you examine the device file for an IDE hard drive, you see this:

```
matthew@seymour:~$ ls -l /dev/sda
brw-rw-- -- 1 root disk 8, 0 2015-06-30 08:13 /dev/sda
```

In this example, `b` designates a block device (a device that transfers and caches data in blocks) with similar permissions. Other device entries you will run across on your Linux system include symbolic links, designated by `s`.

Altering File Permissions with `chmod`

You can use the `chmod` command to alter a file's permissions. This command uses various forms of command syntax, including octal or a mnemonic form (such as `u`, `g`, `o`, or `a` and `rx`, and so on) to specify a desired change. You can use the `chmod` command to add, remove, or modify file or directory permissions to protect, hide, or open up access to a file by other users (except for the root account or a user with super user permission and using `sudo`, either of which can access any file or directory on a Linux system).

The mnemonic forms of `chmod`'s options are (when used with a plus character, `+`, to add, or a minus sign, `-`, to remove):

- ▶ **u**—Adds or removes user (owner) read, write, or execute permission
- ▶ **g**—Adds or removes group read, write, or execute permission
- ▶ **o**—Adds or removes read, write, or execute permission for others not in a file's group
- ▶ **a**—Adds or removes read, write, or execute permission for all users
- ▶ **r**—Adds or removes read permission
- ▶ **w**—Adds or removes write permission
- ▶ **x**—Adds or removes execution permission

For example, if you create a file, such as a `readme.txt`, the file has the following default permissions (set by the `umask` setting in `/etc/bashrc`, covered in the next section):

```
-rw-r--r-- 1 matthew matthew 0 2015-06-30 13:33 readme.txt
```

As you can see, you can read and write the file. Anyone else can only read the file (and only if it is outside your home directory, which will have read, write, and execute permission set only for you, the owner). You can remove all write permission for anyone by using `chmod`, the minus sign (`-`), and `aw`, as follows:

```
matthew@seymour:~$ chmod a-w readme.txt
matthew@seymour:~$ ls -l readme.txt
-r--r--r-- 1 matthew matthew 0 2015-06-30 13:33 readme.txt
```

Now, no one can write to the file (except you, if the file is in your `/home` or `/tmp` directory because of directory permissions). To restore read and write permission for only you as the owner, use the plus sign (`+`) and the `u` and `rw` options, like so:

```
matthew@seymour:~$ chmod u+rw readme.txt
matthew@seymour:~$ ls -l readme.txt
-rw-r--r-- 1 matthew matthew 0 2015-06-30 13:33 readme.txt
```

You can also use the octal form of the `chmod` command (for example, to modify a file's permissions so that only you, the owner, can read and write a file). Use the `chmod` command and a file permission of `600`, like this:

```
matthew@seymour:~$ chmod 600 readme.txt
matthew@seymour:~$ ls -l readme.txt
-rw----- 1 matthew matthew 0 2015-06-30 13:33 readme.txt
```

If you take away execution permission for a directory, files will be hidden inside and may not be listed or accessed by anyone else (except the root operator, of course, who has access to any file on your system). By using various combinations of permission settings, you can quickly and easily set up a more secure environment, even as a normal user in your `/home` directory.

File Permissions with `umask`

When you create a file, it is created with a default set of permissions, `666`. Directories have a default set of permissions, `777`. You can view and modify the default permissions for either with `umask`, which works like a filter. When a file is created by a user account, whether that account is owned by a human like `matthew` or a process like `init`, the file will be created using specific permissions.

The numbers we used earlier when discussing file permissions are also used with `umask`, but with an interesting change. Now, the numbers defined in `umask` are subtracted from the ultimate file permissions. So, if you wanted all new directories to be created with a default permission of `777`, you would type this:

```
matthew@seymour:~$ umask 000
```

Of course, you would never want to have all your directories or files accessible by default because that would be incredibly insecure and unsafe. The default `umask` is `022`, which means that files are created by default with `644` permissions, except in the `/home` directory (for all user directories under it) where the `umask` is `002` and files are created with `775`. Note: The `umask` is a file system-wide variable for each user session, so you cannot have a different setting for specific directories, such as one for `/home` and another for `/home/seymour`.

To find the current `umask` setting, use this:

```
matthew@seymour:~$ umask
```

This may list four digits instead of three. If so, don't be confused. The additional digit is the first one; it is explained later in this chapter, in the section "Understanding Set User ID, Set Group ID, and Sticky Bit Permissions."

To change the `umask` setting—for example, if you wanted the default for files to be `640`—use the following:

```
matthew@seymour:~$ umask 037
```

File Permissions with `chgrp`

You can use the `chgrp` command to change the group to which a file belongs:

```
matthew@seymour:~$ chgrp sudo filename
```

Changing File Permissions with `chown`

You can use the `chown` command to change the owner of a file:

```
matthew@seymour:~$ chown matthew filename
```

You can also use the `chown` command to change the group of a file at the same time:

```
matthew@seymour:~$ chown matthew:sudo filename
```

Understanding Set User ID, Set Group ID, and Sticky Bit Permissions

The first two of the three listed types of permission are "set user ID," known as *suid*, and "set group ID," or *sgid*. These settings, when used in a program, enable any user running that program to have program owner or group owner permissions for that program. These settings enable the program to be run effectively by anyone, without requiring that each user's permissions be altered to include specific permissions for that program.

One commonly used program with *suid* permissions is the `passwd` command:

```
matthew@seymour:~$ ls -l /usr/bin/passwd
-rwsr-xr-x 1 root root 42856 2015-01-26 10:09 /usr/bin/passwd
```

This setting allows normal users to execute the command (as root) to make changes to a root-only-accessible file `/etc/passwd`.

By default, `suid` and `sgid` are turned off on files. To set them, add an extra digit to the beginning of a number in a `chmod` command. `Suid` uses 4. `Sgid` uses 2. You can set both at the same time by using 6 (4 + 2). For example, for a file owned by root with current 711 permissions allowing anyone to run it, you can make it run as root with the following:

```
matthew@seymour:~$ chmod 4711 filename
```

NOTE

Other files that might have `suid` or `guid` permissions include `at`, `rcp`, `rlogin`, `rsh`, `chage`, `chsh`, `ssh`, `crontab`, `sudo`, `sendmail`, `ping`, `mount`, and several UNIX-to-UNIX Copy (UUCP) utilities. Many programs (such as games) might also have this type of permission to access a sound device.

`Sgid` has an additional function when set on directories; in this case, new directory contents are automatically assigned the group owner on the directory.

Files or programs that have `suid` or `guid` permissions can sometimes present security holes because they bypass normal permissions. This problem is compounded if the permission extends to an executable binary (a command) with an inherent security flaw because it could lead to any system user or intruder gaining root access. In past exploits, this typically happened when a user fed a vulnerable command with unexpected input (such as a long pathname or option); the command would fail, and the user would be presented with a root prompt. Although Linux developers are constantly on the lookout for poor programming practices, new exploits are found all the time and can crop up unexpectedly, especially in newer software packages that haven't had the benefit of peer developer review.

Savvy Linux system administrators keep the number of `suid` or `guid` files present on a system to a minimum. The `find` command can be used to display all such files on a system:

```
matthew@seymour:~$ sudo find / -type f -perm /6000 -exec ls -l {} \;
```

NOTE

The `find` command is quite helpful and can be used for many purposes, such as before or during backup operations.

Note that the programs do not necessarily have to be removed from your system. If your users really do not need to use the program, you can remove a program's execute permission for anyone. As the root operator, you have to decide whether your users are allowed, for example, to mount and unmount CD-ROMs or other media on your system. Although Linux-based operating systems can be set up to accommodate ease of use and convenience, allowing programs such as `mount` to be `suid` might not be the best security

policy. Other candidates for `suid` permission change could include the `chsh`, `at`, or `chage` commands.

An additional setting called the *sticky bit* is available using this same additional first digit. A sticky bit limits who may rename or delete files within a directory. When it is set, files in that directory may be unlinked or renamed only by a super user, the directory owner, or the file owner. Set the sticky bit to on by using a `1`, like this for a directory with `755` permissions:

```
matthew@seymour:~$ chmod 1755 directoryname
```

You can set the sticky bit concurrently with `suid` and `sgid`, like this (`4 + 2 + 1`):

```
matthew@seymour:~$ chmod 7755 directoryname
```

Setting Permissions with Access Control Lists

POSIX is a family of standards created to maintain stability and consistency across operating systems for UNIX and UNIX-like systems, such as Linux. One important feature of POSIX is the access control list (ACL; often pronounced “AK-el”). ACLs permit even more fine-grained control over access permissions.

By default, all files have an ACL. To view the ACL for a file, use this:

```
matthew@seymour:~$ getfacl filename
```

Typical `getfacl` output includes multiple lines, like this for `filename.txt`:

```
# file: filename.txt
# owner: matthew
# group: matthew
user::rw-
group::rw-
other::r--
```

The information listed here is standard and clear, based on what you already know. The real power of ACLs is that you can add to them. You are not restricted to the standard set of `user`, `group`, `other`. You can add multiple users and groups with permissions specific to each.

To add the user `sandra` with read, write, and execute permissions to the ACL for a file named `secrets.txt`, use the following:

```
matthew@seymour:~$ setfacl -m u:sandra:rxw secrets.txt
```

To remove and reset `sandra`'s permissions on the file to the file's defaults, use the following:

```
matthew@seymour:~$ setfacl -x u:sandra: secrets.txt
```

From these two examples, you can see that `-m` is for modify and `-r` is for remove.

ACLs permit similar actions with groups and others as with a user. Instead of the `u:` before the name, use a `g:` for groups and an `o:` for others, like this:

```
matthew@seymour:~$ setfacl -m g:groupname:rwx secrets.txt
matthew@seymour:~$ setfacl -m o:r secrets.txt
```

Notice that with others, there is no username or group name to include in the commands.

A useful feature is masking, which allows you to list only the permissions that are available, as in this example:

```
matthew@seymour:~$ setfacl -m m:rx secrets.txt
```

This limits everyone, regardless of any other settings. So, in this case, a group may have `rwx` settings on the file, but the mask here says to permit only `rx`, so `rx` will be the only settings that are available.

As an exercise, see if you can figure out the meaning of this output from `getfacl` for a file named `coffeecup.conf`:

```
# file: coffeecup.conf
# owner: matthew
# group: yirgacheffe
user::rw-
group::rw-
other::r--
group:qa:rwx
group:uat:rwx
mask::rwx
```

Working with Files

Managing files in your home directory involves using one or more easily remembered commands.

Creating a File with `touch`

If you are the user `matthew`, to create an empty file called `myfile` within your current directory, use the following command:

```
matthew@seymour:~$ touch myfile
```

To edit this file, you must use a text editor. (Several text editors are discussed in Chapter 14. However, it is sometimes useful to create an empty file as doing so also creates an access record because of the time and date information that is connected to the file. You can also use `touch` to update this information, called a timestamp, without otherwise accessing or modifying a file.

You can create a file in a different location by changing what is after `touch`. To create a new file in `/home/matthew/randomdirectory`, if you are already in your home directory, you can use the following:

```
matthew@seymour:~$ touch randomdirectory/newfile
```

Or from anywhere using an absolute path, you can use this:

```
matthew@seymour:~$ touch /home/matthew/randomdirectory/newfile
```

Or from anywhere using a path shortcut, you can use the following command:

```
matthew@seymour:~$ touch ~/randomdirectory/newfile
```

Creating a Directory with `mkdir`

To create an empty directory called `newdirectory` within your current directory, use this command:

```
matthew@seymour:~$ mkdir newdirectory
```

If you are the user `matthew`, you can create a directory in a different location by changing what is after `mkdir`. To create a new directory in `/home/matthew/music`, if you are already in your `/home` directory, you can use the following:

```
matthew@seymour:~$ mkdir music/newdirectory
```

Or from anywhere using an absolute path, you can use this:

```
matthew@seymour:~$ mkdir /home/matthew/music/newdirectory
```

Or from anywhere using a path shortcut, you can use the following command:

```
matthew@seymour:~$ mkdir ~/music/newdirectory
```

The `-p` option is valuable. It enables you to create a directory and its parent directories at the same time, if they do not already exist. This can be a real time saver. If the parent directories exist, the command works normally. For example, suppose you want to make a new directory with two layers of subdirectories. In this example, `music` and `newdirectory` already exist, but `subdir1` and `subdir2` are to be created:

```
matthew@seymour:~$ mkdir -p ~/music/newdirectory/subdir1/subdir2
```

Deleting a Directory with `rmdir`

If you are the user `matthew`, to delete an empty directory named `directoryname`, use the following command:

```
matthew@seymour:~$ rmdir directoryname
```


You can remove a directory in a different location by changing what is after `rmdir`. To remove a directory in `/home/matthew/music`, if you are already in your `/home` directory, you can use the following:

```
matthew@seymour:~$ rmdir music/directoryname
```

Or from anywhere using an absolute path, you can use this:

```
matthew@seymour:~$ rmdir /home/matthew/music/directoryname
```

Or from anywhere using a path shortcut, you can use the following command:

```
matthew@seymour:~$ rmdir ~/music/directoryname
```

The directory must be empty to be removed using `rmdir`. However, you can remove a directory with its contents by using `rm`.

CAUTION

You cannot easily recover anything that has been deleted using `rmdir` or `rm`, so proceed carefully. Be absolutely certain you will never need what you are about to delete before you do so. Only a professional data recovery service is likely to be able to recover the files, and even then at great expense.

Deleting a File or Directory with `rm`

If you are the user `matthew`, to delete a file named `filename`, use this command:

```
matthew@seymour:~$ rm filename
```

You can remove a file in a different location by changing what is after `rm`. To remove a directory in `/home/matthew/randomdirectory`, if you are already in your `/home` directory, you can use the following:

```
matthew@seymour:~$ rm randomdirectory/filename
```

Or from anywhere using an absolute path, you can use this:

```
matthew@seymour:~$ rm /home/matthew/randomdirectory/filename
```

Or from anywhere using a path shortcut, you can use the following command:

```
matthew@seymour:~$ rm ~/randomdirectory/filename
```

If you try to use `rm` to remove an empty directory, you receive an error message:

```
rm: cannot remove `random/': Is a directory. In this case, you must use rmdir.
However, you can remove a directory and its contents by using rm.
```

CAUTION

Be sure that all the contents of a directory are known and unwanted if you choose to delete them. There is no way to recover them later. Also, be careful to ensure that you

don't have extra spaces, mistype the name of the directory, or use `sudo` to delete something that you shouldn't be deleting. Linux gives you great power, and it lets you use that power without questioning you about it; that's the human's job.

To delete a directory and all its contents, use the `-r` recursive switch (which works with many commands, not only `rm`):

```
matthew@seymour:~$ rm -r /home/matthew/randomdirectory/
```

Everything in `randomdirectory` as well as the directory itself will be deleted, including other subdirectories, without considering whether they are empty or have contents.

Moving or Renaming a File with `mv`

In Linux land, moving and renaming a file are the same thing. It doesn't matter whether you are moving the directory to another or from one filename to another filename in the same directory; there is only one command to remember. To move a file named `filename` from `~/documents` to `~/archive`, use this command:

```
matthew@seymour:~$ mv documents/filename archive
```

Notice that the filename is not included in the destination. The destination here must be an existing directory. If it is not, the file is renamed to the term used. Some examples will make this clear.

Assuming that you are the user `matthew`, to rename a file that is in your current directory, you could use the following:

```
matthew@seymour:~$ mv oldfilename newfilename
```

To rename a file as you move it from `~/documents` to `~/archive`, you could use this:

```
matthew@seymour:~$ mv documents/oldfilename archive/newfilename
```

Or from anywhere using an absolute path, you could use the following command:

```
matthew@seymour:~$ mv /home/matthew/documents/oldfilename
[ccc]/home/matthew/archive/newfilename
```

Or from anywhere using a path shortcut, you could use this:

```
matthew@seymour:~$ mv ~/documents/oldfilename ~/archive/newfilename
```

Copying a File with `cp`

Copying works similarly to moving, but it retains the original in the original location. Assuming that you are the user `matthew`, to copy a file named `filename` from `~/documents` to `~/archive`, use this command:

```
matthew@seymour:~$ cp documents/filename archive
```

Notice that the filename is not included in the destination. The destination here must be an existing directory. If it is not, the file is renamed to the term used. Some examples will make this clear.

To copy a file that is in your current directory, you could use the following, and it will work exactly the same as `mv`, except that both files will exist afterward:

```
matthew@seymour:~$ cp oldfilename newfilename
```

To rename a file as you copy it from `~/documents` to `~/archive`, you could use this:

```
matthew@seymour:~$ cp documents/oldfilename archive/newfilename
```

Or from anywhere using an absolute path, you could use the following command:

```
matthew@seymour:~$ cp /home/matthew/documents/oldfilename
[ccc]/home/matthew/archive/newfilename
```

Or from anywhere using a path shortcut, you could use this:

```
matthew@seymour:~$ cp ~/documents/oldfilename ~/archive/newfilename
```

Displaying the Contents of a File with `cat`

To view the contents of a text file named `filename` on your screen, assuming that you are the user `matthew`, use this command:

```
matthew@seymour:~$ cat filename
```

Notice that the text is displayed on your screen but that you cannot edit or work with the text in any way. This command is convenient when you want to know the contents of a file but don't need to make any changes. Text editors for the terminal are covered in Chapter 12. This command works best with short files because the contents of longer files scroll off the screen too quickly to be read.

Displaying the Contents of a File with `less`

When you need to view the contents of a longer text file from the command line, you can use `less`. This produces a paged output, meaning that output stops each time your screen is full. You can then use your up- and down-arrow keys and page-up and page-down keys to scroll through the contents of the file. Then, use `q` to quit and return to the command line:

```
matthew@seymour:~$ less filename
```

In the early days of UNIX, a program called `more` gave paged output. It was the first paged output program but did not include the ability to scroll up and down. `less` was written to add that capability and was named as a bit of hacker humor because “less is more.”

Using Wildcards and Regular Expressions

Each of the commands in the previous sections can be used with pattern-matching strings known as *wildcards* or *regular expressions*. For example, to delete all files in the current directory beginning with the letters `abc`, you can use an expression beginning with the first three letters of the desired filenames. An asterisk (`*`) is then appended to match all these files. Use a command line with the `rm` command like this:

```
matthew@seymour:~$ rm abc*
```

Linux shells recognize many types of file-naming wildcards, but this is different from the capabilities of Linux commands supporting the use of more complex expressions. You learn more about using wildcards in Chapter 11 and in Chapter 14.

NOTE

You can also learn more about using expressions by reading the `grep` manual pages (`man grep`), but because both `man` and `grep` are covered in Chapter 11, this mention is included only to whet your appetite.

Working as Root

The root, or super user, account is a special account and user on UNIX and Linux systems. Super user permissions are required in part because of the restrictive file permissions assigned to important system configuration files. You must have root permission to edit these files or to access or modify certain devices (such as hard drives). When logged in as root, you have total control over your system, which can be dangerous.

When you work in root, you can destroy a running system with a simple invocation of the `rm` command like this:

```
matthew@seymour:~$ sudo rm -rf / --no-preserve-root
```

This command line not only deletes files and directories but also could wipe out file systems on other partitions and even remote computers. This alone is reason enough to take precautions when using root access.

The only time you should run Linux as the super user is when you are configuring the file system, for example, or to repair or maintain the system. Logging in and using Linux as the root operator isn't a good idea because it defeats the entire concept of file permissions.

Knowing how to run commands as the super user (root) without logging in as root can help avoid serious missteps when configuring your system. In Ubuntu, you can use `sudo` to execute single commands as root and then immediately return to normal user status. For example, if you would like to edit your system's file system table (a text file that describes local or remote storage devices, their type, and location), you can use `sudo` like this:

```
matthew@seymour:~$ sudo nano -w /etc/fstab
[sudo] password for matthew:
```

After you press Enter, you are prompted for a password that gives you access to root. This extra step can also help you think before you leap into the command. Enter the root password, and you are then editing `/etc/fstab`, using the `nano` editor with line wrapping disabled (thanks to the `-w`).

CAUTION

Before editing any important system or software service configuration file, make a backup copy. Then make sure to launch your text editor with line wrapping disabled. If you edit a configuration file without disabling line wrapping, you could insert spurious carriage returns and line feeds into its contents, causing the configured service to fail when restarting. By convention, nearly all configuration files are formatted for 80-character text width, but this is not always the case. By default, the `vi` and `emacs` editors don't use line wrapping.

Understanding and Fixing `sudo`

Most Ubuntu users never have problems with `sudo`, but sometimes, people who like to experiment break things, especially while learning. This section helps you understand more completely how `sudo` works and also how to restore `sudo` access to a specific user when, for some reason, it has ceased to function for that user.

NOTE

You usually can tell that a problem has occurred because an error message like this appears when a user tries to issue a command using `sudo`:

```
matthew@seymour:~$ sudo shutdown -h now
[sudo] password for matthew:
matthew is not in the sudoers file. This incident will be reported.
```

Sometimes, you might not even receive an error message, but the command issued simply does nothing. Either way, you can fix the problem by using the following knowledge and procedure.

In order for a user to use `sudo`, the user account must belong to the `sudo` group and must also be listed in the `/etc/sudoers` file. If both conditions are met, the user will be permitted to temporarily use root powers for specific commands that are issued at the command line by that user account by prefacing the command with the word `sudo`.

A problem can occur for a specific user with `sudo` in several situations:

- ▶ When the user is taken out of the `sudo` group but should not have been
- ▶ When the permissions for the `/etc/sudoers` file have been changed to anything other than `440`
- ▶ When the `/etc/sudoers` file has been changed in a way that does not allow members of the `sudo` group to use root powers

These problems generally result from users doing something they should not have done, such as changing the permissions on all files rather than taking the time to figure out a specific file that is causing problems due to permissions issues. Take heed: It is better to spend a bit more time learning than it is to take a shortcut that causes bigger problems.

Fixing any of these problems requires the use of root powers. This is an obvious problem because if `sudo` is not working, the account does not have access to root. To fix it, you must gain root access. You can do this by booting into *recovery mode*, using the following steps:

1. Hold down the Shift key while the computer is booting.
2. When the GRUB menu page appears, use the arrow keys on your keyboard to scroll to the entry that ends with *(recovery mode)* and press Enter to select it.
3. When the boot process finishes, and have several options, select the menu entry for root: Drop to Root Shell Prompt. You are now at the command line, with full root access to the computer.
4. Because Ubuntu mounts file systems as read-only by default in recovery mode, you need to remount the root file system, `/`, as read/write so that you can fix the problem. Enter the following:

```
root@seymour:~# mount -o rw,remount /
```

NOTE

You now have complete root access and read/write privileges on the machine. This is an example of why security of a physical machine is important. If someone has physical access to your computer and knows what he or she is doing, that person can easily and quickly gain full control over the machine and all it contains.

If the problem exists because the user account was removed from the admin group, enter the following:

```
root@seymour:~# adduser username admin
```

If the problem exists because the permissions for `/etc/sudoers` are wrong, enter this:

```
root@seymour:~# chmod 440 /etc/sudoers
```

If the problem exists because of an internal problem in `/etc/sudoers`, make a backup of the existing file and use `visudo` to edit it. (This is a special use of the `vi` editor, covered in Chapter 12, that runs a check on the file after editing to be certain it is correct; this particular problem usually occurs when someone edits the file using another editor that does not make this check.) The contents of the file should be as follows:

```
#
# This file MUST be edited with the 'visudo' command as root.
#
# Please consider adding local content in /etc/sudoers.d/ instead of
# directly modifying this file.
```

```

#
# See the man page for details on how to write a sudoers file.
#
Defaults env_reset
Defaults secure_path="/usr/local/sbin:/usr/local/bin:/usr/sbin:/usr/bin:/sbin:/bin"

# Host alias specification

# User alias specification

# Cmnd alias specification

# User privilege specification
root ALL=(ALL:ALL) ALL

# Members of the admin group may gain root privileges
%admin ALL=(ALL) ALL

# Allow members of group sudo to execute any command
%sudo ALL=(ALL:ALL) ALL

# See sudoers(5) for more information on "#include" directives:

#include_dir /etc/sudoers.d

```

After your fix is complete, exit the root command line:

```
root@seymour:~# exit
```

You return to the recovery mode menu. Select Resume Normal Boot to finish and return to a normal boot. When the boot completes, you should be able to use `sudo` correctly again.

Creating Users

When a Linux system administrator creates a user, an entry is created in `/etc/passwd` for the user. The system also creates a directory, labeled with the user's username, in the `/home` directory. For example, if you create a user named `sandra`, the user's home directory is `/home/sandra`.

NOTE

In this chapter, you learn how to manage users from the command line. See Chapter 13, "Managing Users," for more information on user administration, including administration using graphical administration utilities.

Use the `adduser` command, along with a user's name, to quickly create a user:

```
matthew@seymour:~$ sudo adduser sandra
```

After creating a user, you must also create the user's initial password with the `passwd` command:

```
matthew@seymour:~$ sudo passwd sandra
```

```
Changing password for user sandra.
```

```
New password:
```

```
Retype new password:
```

```
passwd: all authentication tokens updated successfully.
```

Enter the new password twice. If you do not create an initial password for a new user, the user cannot log in.

The `adduser` command has many command-line options. The command can be used to set policies and dates for the new user's password, assign a login shell, assign group membership, and other aspects of a user's account. See `man adduser` as well as Chapter 13 for more info.

Deleting Users

Use the `deluser` command to delete users from your system. This command removes a user's entry in the system's `/etc/passwd` file. You should also use the command's `--remove-all-files` and `--remove-home` options to remove all the user's files and directories (such as the user's mail spool file under `/var/spool/mail`):

```
matthew@seymour:~$ sudo deluser --remove-all-files --remove-home andrew
```

If you do not use the `--remove-home` option, you have to manually delete the user's directory under `/home`, along with the user's `/var/spool/mail` queue.

Shutting Down the System

Use the `shutdown` command to shut down your system. The `shutdown` command has a number of different command-line options (such as shutting down at a predetermined time), but the fastest way to cleanly shut down Linux is to use the `-h` (or `halt`) option, followed by the word `now`:

```
matthew@seymour:~$ sudo shutdown -h now
```

You can also follow `-h` with the numeral zero (0), like this, to get the same effect:

```
matthew@seymour:~$ sudo shutdown -h 0
```

To incorporate a timed shutdown and a pertinent message to all active users, use `shutdown`'s time and message options, as follows:

```
matthew@seymour:~$ sudo shutdown -h 18:30 "System is going down for maintenance
this evening at 6:30 p.m. Please make sure you have saved your work and logged
out by then or you may lose data."
```


This example shuts down your system and provides a warning to all active users 15 minutes before the shutdown (or reboot). Shutting down a running server can be considered drastic, especially if there are active users or exchanges of important data occurring (such as a backup in progress). One good approach is to warn users ahead of time. This can be done by editing the system *Message of the Day (MOTD)* `motd` file, which displays a message to users when they log in using the command-line interface, as is common on multiuser systems.

It used to be that to create a custom MOTD file, you only had to use a text editor and change the contents of `/etc/motd`. However, this has changed in Ubuntu, as the developers have added a way to automatically and regularly update some useful information contained in the MOTD file by using `cron`. To modify how the MOTD file is updated, you should install `update-motd` and read the man page.

You can also make downtimes part of a regular schedule, perhaps to coincide with security audits, software updates, or hardware maintenance.

You should shut down Ubuntu for only a few very specific reasons:

- ▶ You are not using the computer, no other users are logged in or expected to need or use the system (as with your personal desktop or laptop computer), and you want to conserve electrical power.
- ▶ You need to perform system maintenance that requires any or all system services to be stopped.
- ▶ You want to replace integral hardware.

TIP

Do not shut down your computer if you suspect that intruders have infiltrated your system; instead, disconnect the machine from any or all networks and make a backup copy of your hard drives. You might want to also keep the machine running to examine the contents of memory and to examine system logs. Exceptions to this are when the system contains only trivial data files and nonessential services, such as a personal computer that is only used to run a web browser, and when you have no intention of trying to track down what an intruder might have changed, either to repair the damage or to try to catch the intruder by using computer forensics, but rather plan to merely wipe everything clean and rebuild or reinstall the system from scratch.

Rebooting the System

You should use the `shutdown` command to reboot your system. The fastest way to cleanly reboot Linux is to use the `-r` option and the word `now`:

```
matthew@seymour:~$ sudo shutdown -r now
```

You can also follow `-r` with the numeral zero (0), like this, to get the same effect:

```
matthew@seymour:~$ sudo shutdown -r 0
```

Both rebooting and shutting down can have dire consequences if performed at the wrong time (such as during backups or critical file transfers, which arouses the ire of your system's users). However, Linux-based operating systems are designed to properly stop active system services in an orderly fashion. Other commands you can use to shut down and reboot Linux are the `halt`, `poweroff`, and `reboot` commands, but the `shutdown` command is more flexible.

Commonly Used Commands and Programs

A number of programs and built-in shell commands are commonly used when working at the command line. These commands are organized here by category to help you understand the purpose of each category:

- ▶ **Managing users and groups**—`chage`, `chfn`, `chsh`, `edquota`, `gpasswd`, `groupadd`, `groupdel`, `groupmod`, `groups`, `mkpasswd`, `newgrp`, `newusers`, `passwd`, `umask`, `useradd`, `userdel`, `usermod`
- ▶ **Managing files and file systems**—`cat`, `cd`, `chattr`, `chmod`, `chown`, `compress`, `cp`, `dd`, `fdisk`, `find`, `gzip`, `ln`, `mkdir`, `mksfs`, `mount`, `mv`, `rm`, `rmdir`, `rpm`, `sort`, `swapon`, `swapoff`, `tar`, `touch`, `umount`, `uncompress`, `uniq`, `unzip`, `zip`
- ▶ **Managing running programs**—`bg`, `fg`, `kill`, `killall`, `nice`, `ps`, `pstree`, `renice`, `top`, `watch`
- ▶ **Getting information**—`apropos`, `cal`, `cat`, `cmp`, `date`, `diff`, `df`, `dir`, `dmesg`, `du`, `env`, `file`, `free`, `grep`, `head`, `info`, `last`, `less`, `locate`, `ls`, `lsattr`, `man`, `more`, `pinfo`, `ps`, `pwd`, `stat`, `strings`, `tac`, `tail`, `top`, `uname`, `uptime`, `vdir`, `vmstat`, `w`, `wc`, `whatis`, `whereis`, `which`, `who`, `whoami`
- ▶ **Console text editors**—`ed`, `jed`, `joe`, `mcedit`, `nano`, `red`, `sed`, `vim`
- ▶ **Console Internet and network commands**—`bing`, `elm`, `ftp`, `host`, `hostname`, `ifconfig`, `links`, `lynx`, `mail`, `mutt`, `ncftp`, `netconfig`, `netstat`, `pine`, `ping`, `pump`, `rdate`, `route`, `scp`, `sftp`, `ssh`, `tcpdump`, `traceroute`, `whois`, `wire-test`

If you need to find full information for using the command, you can find that information under the command's man page.

References

- ▶ <https://help.ubuntu.com/community/UsingTheTerminal>—The Ubuntu community help page for using the terminal
- ▶ <https://help.ubuntu.com/community/LinuxFilesystemTreeOverview>—The Ubuntu community help page for and overview of the Linux file system tree
- ▶ <https://help.ubuntu.com/community/RootSudo>—An Ubuntu community page explaining `sudo`, the philosophy behind using it by default, and how to use it

This page intentionally left blank

CHAPTER 11

Command-Line Master Class, Part 1

Some Linux users like to focus on the graphical environments that are available; they rush to tell new users that the command line isn't vital when using Linux. Although there are some amazing *graphical user interface (GUI)* desktops, and this statement is mostly true, avoiding the command line limits your options and makes some tasks more difficult. The command-line interface offers the greatest power and flexibility, and those who actively avoid learning how to use it are also actively limiting their abilities and options. You learned the basics in Chapter 10, "Command-Line Beginner's Class." In this chapter, we dig in deeper.

It is with some trepidation that we retain the classic title "Command-Line Master Class" for this two-chapter set. Entire books have been published covering the depth and breadth of the command line. To believe that two short chapters make any reader a true master is foolish. Our greatest hope is to give enough information to enable any reader to perform all basic and vital tasks from the command line while inspiring readers to go on a quest to discover all the beauty and grandeur that we do not have space to cover here. Please keep this in mind as you continue.

In his book *The Art of Unix Programming*, Eric Raymond wrote a short story that perfectly illustrates the power of the command line versus the GUI. It's reprinted here with permission, for your reading pleasure:

One evening, Master Foo and Nubi attended a gathering of programmers who had met to learn from each other. One of the programmers asked Nubi to what school he and his master belonged. Upon being told they were followers of the Great Way of Unix, the programmer grew scornful.

IN THIS CHAPTER

- ▶ Why Use the Command Line?
- ▶ Using Basic Commands
- ▶ References

“The command-line tools of Unix are crude and backward,” he scoffed.

“Modern, properly designed operating systems do everything through a graphical user interface.”

Master Foo said nothing, but pointed at the moon. A nearby dog began to bark at the master’s hand.

“I don’t understand you!” said the programmer.

Master Foo remained silent, and pointed at an image of the Buddha. Then he pointed at a window. “What are you trying to tell me?” asked the programmer.

Master Foo pointed at the programmer’s head. Then he pointed at a rock.

“Why can’t you make yourself clear?” demanded the programmer.

Master Foo frowned thoughtfully, tapped the programmer twice on the nose, and dropped him in a nearby trash can.

As the programmer was attempting to extricate himself from the garbage, the dog wandered over and piddled on him.

At that moment, the programmer achieved enlightenment.

Whimsical as the story is, it does illustrate that there are some things that the GUI just does not do well. Enter the command line: It is a powerful and flexible operating environment and—if you practice—can actually be quite fun, too!

In this chapter, you learn more commands to help you master the command line so that you can perform common tasks through it.

Why Use the Command Line?

Moving from the GUI to the command line is a conscious choice for most people, although it is increasingly rare that it is an absolute choice accompanied by complete abandonment of GUIs.

Reasons for using the command line include the following:

- ▶ You want to chain together two or more commands.
- ▶ You want to use a command or parameter available only on the shell.
- ▶ You are working on a text-only system.
- ▶ You have used it for a long time and feel comfortable there.
- ▶ You want to automate a task.

Chaining together two or more commands, or *piping*, is what gives the shell its real power. Hundreds of commands are available, and by combining them in different ways, you get tons of new options. Some of the shell commands are available through the GUI, but

these commands usually have only a small subset of their parameters available, which limits what you can do with them.

Working from a text-only system is useful both for working locally with a broken GUI and for connecting to a remote, text-only system. If your Linux server is experiencing problems, the last thing you want to do is load it down with a GUI connection; working in text mode is faster and more efficient.

Many people use the shell simply because it is familiar to them. Some people even use the shell to start GUI applications just because it saves them taking their hands off the keyboard for a moment. This is not a bad thing; it provides fluency and ease with the system and is a perfectly valid way of working. Working from the command line is faster. The mouse is slow, and taking your fingers away from the keyboard makes your work even slower. Anyone looking to achieve the Zen-like power user state hinted at by Eric Raymond will understand this after making the effort to learn.

Knowing how to work in the shell is also essential if you want to automate tasks on your system without use of the GUI. Whether you want to add a periodic task to `cron` or update a configuration management system, you need to know what text commands to give to run programs from the command line.

Using Basic Commands

It is impossible to know how many commands the average command-line citizen uses, but if we had to guess, we would place it at about 25. Some of these were introduced in Chapter 10 but are covered here in greater depth. Others may be new to you. Still others are mentioned in this list only to provide ideas for further study. Here are some commands that every command-line user will want to learn:

- ▶ `cat`—Prints the contents of a file
- ▶ `cd`—Changes directories
- ▶ `chmod`—Changes file access permissions
- ▶ `cp`—Copies files
- ▶ `du`—Prints disk usage
- ▶ `emacs`—Edits text files
- ▶ `find`—Finds files by searching
- ▶ `grep`—Searches for a string in input or files
- ▶ `head`—Prints the first lines of a file
- ▶ `less`—Displays files or input interactively
- ▶ `ln`—Creates links between files
- ▶ `locate`—Finds files from an index
- ▶ `ls`—Lists files in the current directory

- ▶ **make**—Compiles and installs programs
- ▶ **man**—Displays manual pages for reading
- ▶ **mkdir**—Makes directories
- ▶ **mv**—Moves files
- ▶ **nano**—Edits text files
- ▶ **rm**—Deletes files and directories
- ▶ **sort**—Takes a text file as input and outputs the contents of the file in the order you specify
- ▶ **ssh**—Connects to other machines using a secure shell connection
- ▶ **tail**—Prints the last lines of a file
- ▶ **vim**—Edits text files
- ▶ **which**—Prints the location of a command

Many other commands are also used fairly often—`cut`, `diff`, `gzip`, `history`, `ping`, `su`, `tar`, `uptime`, `who`, and so on—but if you can understand the ones listed here, you have sufficient skill to concoct your own command combinations.

Note that we say *understand* the commands—not know all their possible parameters and usages. This is because several of the commands, although commonly used, are used only in any complex manner by people with specific needs. `make` is a good example of this: Unless you plan to become a programmer, you need not worry about this command beyond just using `make` and `make install` now and then. If you want to learn more, see Chapter 38, “Using Programming Tools.”

Similarly, `emacs`, `nano`, and `vim` are text editors that have text-based interfaces all their own and are covered later in this chapter. `ssh` is covered in detail in Chapter 19, “Remote Access with SSH and VNC.”

The rest of this list is composed of commands that each have many parameters you can use to customize what the commands actually do. Again, many of the parameters are esoteric and rarely used, and the few times in your Linux life that you need them, you can just read the manual page.

We go over these commands one by one, explaining the most common ways to use them.

Printing the Contents of a File with `cat`

Many of Ubuntu’s shell commands manipulate text strings, so if you want to be able to feed them the contents of files, you need to be able to output those files as text. Enter the `cat` command, which prints the contents of any files you pass to it.

Its most basic use is like this:

```
matthew@seymour:~$ cat myfile.txt
```

This prints the contents of `myfile.txt`. For this usage, two extra parameters are often used: `-n` numbers the lines in the output, and `-s` (“squeeze”) prints a maximum of one blank line at a time. That is, if your file has 1 line of text, 10 blank lines, 1 line of text, 10 blank lines, and so on, `-s` shows the first line of text, a single blank line, the next line of text, a single blank line, and so forth. When you combine `-s` and `-n`, `cat` numbers only the lines that are printed—the 10 blank lines shown as 1 will count as 1 line for numbering.

Assuming that you are the user `matthew`, the following command prints information about your CPU, stripping out multiple blank lines and numbering the output:

```
matthew@seymour:~$ cat -sn /proc/cpuinfo
```

You can also use `cat` to print the contents of several files at once, like this:

```
matthew@seymour:~$ cat -s myfile.txt myotherfile.txt
```

In this command, `cat` merges `myfile.txt` and `myotherfile.txt` on the output and strips out multiple blank lines. The important thing is that `cat` does not distinguish between the files in the output; there are no filenames printed and no extra breaks between the two. This allows you to treat the 2 files as 1 or, by adding more files to the command line, to treat 20 files as 1.

Changing Directories with `cd`

Changing directories is surely something that has no options, right? Well, `cd` is actually more flexible than most people realize. Unlike most of the other commands here, `cd` is not a command in itself—it is built in to `bash` (or whichever shell interpreter you are using), but it is still used like a command.

The most basic usage of `cd` is this:

```
matthew@seymour:~$ cd somedir
```

This looks in the current directory for the `somedir` subdirectory and then moves you into it. You can also specify an exact location for a directory, like this:

```
matthew@seymour:~$ cd /home/matthew/stuff/somedir
```

The first part of `cd`’s magic lies in the characters (`-` and `~`, a dash and a tilde). The first means “switch to my previous directory,” and the second means “switch to my home directory.” The following conversation with `cd` shows this in action:

```
matthew@seymour:~$ cd /usr/local
matthew@seymour/usr/local$ cd bin
matthew@seymour/usr/local/bin$ cd -
/usr/local
matthew@seymour/usr/local$ cd ~
matthew@seymour:~$
```


In the first line, you change to `/usr/local` and get no output from the command. In the second line, you change to `bin`, which is a subdirectory of `/usr/local`. Next, `cd -` is used to change back to the previous directory. This time `bash` prints the name of the previous directory so you know where you are. Finally, `cd ~` is used to change back to your `/home/matthew` directory, although if you want to save an extra few keystrokes, you can just type `cd` by itself, which is equivalent to `cd ~`.

The second part of `cd`'s magic is its capability to look for directories in predefined locations. When you specify an absolute path to a directory (that is, one starting with a `/`), `cd` always switches to that exact location. However, if you specify a relative subdirectory—for example, `cd subdir`—you can tell `cd` where you would like that to be relative to. This is accomplished with the `CDPATH` environment variable. If this variable is not set, `cd` always uses the current directory as the base; however, you can set it to any number of other directories.

This next example shows a test of this. It starts in `/home/matthew/empty`, an empty directory, and the lines are numbered for later reference:

```
1 matthew@seymour:~/empty$ pwd
2 /home/matthew/empty
3 matthew@seymour:~/empty$ ls
4 matthew@seymour:~/empty$ mkdir local
5 matthew@seymour:~/empty$ ls
6 local
7 matthew@seymour:~/empty$ cd local
8 matthew@seymour:~/empty/local$ cd ..
9 matthew@seymour:~/empty$ export CDPATH=/usr
10 matthew@seymour:~/empty$ cd local
11 /usr/local
12 matthew@seymour:/usr/local$ cd -
13 /home/matthew/empty
14 matthew@seymour:~/empty$ export CDPATH=./usr
15 matthew@seymour:~/empty$ cd local
16 /home/matthew/empty/local
17 matthew@seymour:~/empty/local$
```

Lines 1–3 show that you are in `/home/matthew/empty` and that it is indeed empty; `ls` had no output. Lines 4–6 show the `local` subdirectory being made, so that `/home/matthew/empty/local` exists. Lines 7 and 8 show that you can `cd` into `/home/matthew/empty/local` and back out again.

In line 9, `CDPATH` is set to `/usr`. This was chosen because Ubuntu has the directory `/usr/local`, which means your current directory (`/home/matthew/empty`) and your `CDPATH` directory (`/usr`) both have local subdirectories. In line 10, while in the `/home/matthew/empty` directory, you use `cd local`. This time, `bash` switches you to `/usr/local` and even prints the new directory to ensure that you know what it has done.

Lines 12 and 13 move you back to the previous directory, `/home/matthew/empty`. In line 14, `CDPATH` is set to be `./usr`. The `.` is the directory separator, so this means `bash` should look first in the current directory, `.`, and then in the `/usr` directory. In line 15 `cd`

`local` is issued again, this time moving to `/home/matthew/empty/local`. Note that `bash` has still printed the new directory; it does that whenever it looks up a directory in `CDPATH`.

Changing File Access Permissions with `chmod`

Your use of `chmod` can be greatly extended through one simple parameter: `-c`. This instructs `chmod` to print a list of all the changes it made as part of its operation, which means you can capture the output and use it for other purposes. Consider this example:

```
matthew@seymour:~$ chmod -c 600 *
mode of '1.txt' changed to 0600 (rw-----)
mode of '2.txt' changed to 0600 (rw-----)
mode of '3.txt' changed to 0600 (rw-----)
matthew@seymour:~$ chmod -c 600 *
matthew@seymour:~$
```

Here the `chmod` command is issued with `-c`, and you can see it has output the result of the operation: Three files were changed to `rw-----` (read and write by user only). However, when the command is issued again, no output is returned. This is because `-c` prints only the changes it made. Files that already match the permissions you are setting are left unchanged and therefore are not printed.

Two other parameters of interest are `--reference` and `-R`. The first allows you to specify a file to use as a template for permissions rather than specifying permissions yourself. For example, if you want all files in the current directory to have the same permissions as the file `/home/matthew/myfile.txt`, you use this:

```
chmod --reference /home/matthew/myfile.txt *
```

You can use `-R` to enable recursive operation, which means you can use it to `chmod` a directory, and it will change the permissions of that directory as well as all files and subdirectories under that directory. You could use `chmod -R 600 /home` to change every file and directory under `/home` to become read/write to its owner(s).

Copying Files with `cp`

Like `mv`, which is covered later in this chapter, `cp` is a command that is easily used and mastered. However, two marvelous parameters rarely see much use, despite their power—which is a shame. These are `--parents` and `-u`. The first copies the full path of the file into the new directory; the second copies only if the source file is newer than the destination.

Using `--parents` requires a little explanation, so here is an example. Say that you have a file, `/home/matthew/desktop/documents/work/notes.txt`, and want to copy it to your `/home/matthew/backup` folder. You could do a normal `cp`, but that would give you `/home/matthew/backup/notes.txt`, so how would you know where that came from later? If you use `--parents`, the file is copied to `/home/matthew/backup/desktop/documents/work/notes.txt`.

The `-u` parameter is perfect for synchronizing two directories because it allows you to run a command like `cp -Ru myfiles myotherfiles` and have `cp` recopy only files that have changed. The `-R` parameter means *recursive* and enables you to copy directory contents.

Printing Disk Usage with `du`

The `du` command prints the size of each file and directory that is inside the current directory. Its most basic usage is as easy as it gets:

```
matthew@seymour:~$ du
```

This outputs a long list of directories and how much space their files take up. You can modify it with the `-a` parameter, which instructs `du` to print the sizes of individual files as well as directories. Another useful parameter is `-h`, which makes `du` use human-readable sizes like 18M (18MB) rather than 17532 (the same number in bytes, unrounded). The final useful basic option is `-c`, which prints the total sizes of files.

So, by using `du` you can get a printout of the size of each file in your `/home` directory, in human-readable format, and with a summary at the end, like this:

```
matthew@seymour:~$ du -ahc /home/matthew
```

Two advanced parameters deal with filenames you want excluded from your count. The first is `--exclude`, which enables you to specify a pattern that should be used to exclude files. This pattern is a standard shell file-matching pattern as opposed to a regular expression, which means you can use `?` to match a single character or `*` to match zero or many characters. You can specify multiple `--exclude` parameters to exclude several patterns. Here is an example:

```
matthew@seymour:~$ du --exclude="*.xml" --exclude="*.xsl"
```

However, typing numerous `--exclude` parameters repeatedly is a waste of time, and you can use `-x` to specify a file that has the list of patterns you want excluded. The file should look like this:

```
*.xml
*.xsl
```

That is, each pattern you want excluded should be on a line by itself. If that file were called `xml_exclude.txt`, you could use it in place of the previous example, like this:

```
matthew@seymour:~$ du -X xml_exclude.txt
```

You can make your exclusion file as long as you need, or you can just specify multiple `-x` parameters.

TIP

If you run `du` in a directory where several files are hard-linked to the same inode, you count the size of the file only once. If you want to count each hard link separately for some reason, use the `-l` parameter (lowercase *L*).

Using `echo`

You can do many things with `echo`, especially with redirection (see Chapter 12, “Command-Line Master Class, Part 2,” for more about redirecting output). In its simplest use, `echo` sends whatever you tell it to send to standard output. If you want to repeat text on

the screen (which is useful in a shell script, for example), just enter the text string in single quotation marks ('), and the output appears below it, like this:

```
matthew@seymour:~$ echo 'I have the power!'
I have the power!
```

If you want to know the value of a system variable, such as `TERM`, enter the variable name to output the value, like this:

```
matthew@seymour:~$ echo $TERM
xterm
```

You can redirect the output of `echo` into a text file, as is done here to add a new directory to `PATHS`:

```
matthew@seymour:~$ echo 'export PATH=$PATH:/usr/local/bin' >> ~/.bashrc
```

You can change or set a kernel setting (1 = on, 0 = off) in `/proc`, like this:

```
matthew@seymour:~$ sudo sh -c 'echo "1" > /proc/sys/location/of/setting'
```

Note that you can read the setting of a kernel value in `/proc` by using `cat`.

Finding Files by Searching with `find`

The `find` command is one of the darkest and least understood areas of Linux, but it is also one of the most powerful. Admittedly, the `find` command does not help itself by using X-style parameters. The UNIX standard is `-c`, `-s`, and so on, whereas the GNU standard is `--dosomething`, `--mooby`, and so forth. X-style parameters merge the two by having words preceded by only one dash.

However, the biggest problem with `find` is that it has more options than most people can remember; it truly is capable of doing most things you could want. The most basic usage is as follows:

```
matthew@seymour:~$ find -name "*.txt"
```

This option searches the current directory and all subdirectories for files that end in `.txt`. The previous search finds files ending in `.txt` but not `.TXT`, `.Ttxt`, or other case variations. To search without case-sensitivity, use `-iname` rather than `-name`. You can optionally specify where the search should start before the `-name` parameter, like this:

```
matthew@seymour:~$ find /home -name "*.txt"
```

Another useful test is `-size`, which you use to specify how big the files should be in order to match. You can specify the size in kilobytes and optionally use `+` or `-` to specify greater than or less than. Consider these examples:

```
matthew@seymour:~$ find /home -name "*.txt" -size 100k
matthew@seymour:~$ find /home -name "*.txt" -size +100k
matthew@seymour:~$ find /home -name "*.txt" -size -100k
```

The first example brings up files of exactly 100KB, the second only files larger than 100KB, and the last only files under 100KB.

Moving on, the `-user` option enables you to specify the user who owns the files you are looking for. So, to search for all files in `/home` that end with `.txt`, that are under 100KB, and that are owned by user `matthew`, you use this:

```
matthew@seymour:~$ find /home -name "*.txt" -size -100k -user matthew
```

You can flip any of the conditions by specifying `-not` before them. For example, you can add `-not` before `-user matthew` to find matching files owned by everyone except `matthew`:

```
matthew@seymour:~$ find /home -name "*.txt" -size -100k -not -user matthew
```

You can add as many `-not` parameters as you need, and you can even use `-not -not` to cancel out the first `-not`. (Yes, that is pointless.) Keep in mind, though, that `-not -size -100k` is essentially equivalent to `-size +100k`, with the exception that the former will match files of exactly 100KB, whereas the latter will not.

You can use `-perm` to specify which permissions a file should have in order to match it. This is tricky, so read carefully. The permissions are specified in the same way as with the `chmod` command: `u` for user, `g` for group, `o` for others, `r` for read, `w` for write, and `x` for execute. However, before you give the permissions, you need to specify a plus, a minus, or a blank space. If you specify neither a plus nor a minus, the files must exactly match the mode you give. If you specify `-`, the files must match all the modes you specify. If you specify `+`, the files must match any the modes you specify. Confused yet?

The confusion can be cleared up with some examples. This following command finds all files that have permission `o=r` (readable for other users). Notice that if you remove the `-name` parameter, it is equivalent to `*` because all filenames are matched:

```
matthew@seymour:~$ find /home -perm -o=r
```

Any files that have `o=r` set are returned from this query. Those files also might have `u=rw` and other permissions, but as long as they have `o=r`, they will match. This next query matches all files that have `o=rw` set:

```
matthew@seymour:~$ find /home -perm -o=rw
```

However, this query does not match files that are `o=r` or `o=w`. To be matched, a file must be readable and writable by other users. If you want to match readable or writable (or both), you need to use `+`, like this:

```
matthew@seymour:~$ find /home -perm +o=rw
```

Similarly, the next query matches only files that are readable by the user, the group, and others:

```
matthew@seymour:~$ find /home -perm -ugo=r
```

On the other hand, the following query matches files as long as they are readable by the user, or by the group, or by others, or by any combination of the three:

```
matthew@seymour:~$ find /home -perm +ugo=r
```

If you use neither `+` nor `-`, you are specifying the exact permissions to search for. For example, the following query searches for files that are readable by the user, the group, and others but not writable or executable by anyone:

```
matthew@seymour:~$ find /home -perm ugo=r
```

You can be as specific as you need to be with the permissions. For example, this query finds all files that are readable for the user, the group, and others and writable by the user:

```
matthew@seymour:~$ find /home -perm ugo=r,u=w
```

To find files that are not readable by others, use the `-not` condition, like this:

```
matthew@seymour:~$ find /home -not -perm +o=r
```

Now, on to the most advanced aspect of the `find` command: the `-exec` parameter. This parameter enables you to execute an external program each time a match is made, passing in the name of the matched file wherever you want it. This has very specific syntax: Your command and its parameters should follow immediately after `-exec`, terminated by `\;`. You can insert the filename match at any point by using `{}` (an opening and a closing brace side by side).

So, you can match all text files on the entire system (that is, searching recursively from `/` rather than from `/home`, as in the earlier examples) over 10KB, owned by `matthew`, that are not readable by other users, and then use `chmod` to enable reading, like this:

```
matthew@seymour:~$ find / -name "*.txt" -size +10k -user matthew -not -perm +o=r
-exec chmod o+r {} \;
```

When you type your own `-exec` parameters, be sure to include a space before `\;`. Otherwise, you might see an error such as `missing argument to '-exec'`.

Do you see now why some people think the `find` command is scary? Many people learn just enough about `find` to be able to use it in a very basic way, but hopefully you will see how much it can do if you give it chance.

Searches for a String in Input with `grep`

The `grep` command, like `find`, is an incredibly powerful search tool in the right hands. Unlike `find`, though, `grep` processes any text, whether in files or just in standard input.

The basic usage of `grep` is as follows:

```
matthew@seymour:~$ grep "some text" *
```

This searches all files in the current directory (but not subdirectories) for the string `some text` and prints matching lines along with the name of the file. To enable recursive searching in subdirectories, use the `-r` parameter, as follows:

```
matthew@seymour:~$ grep -r "some text" *
```

Each time a string is matched within a file, the filename and the match are printed. If a file contains multiple matches, each of the matches is printed. You can alter this behavior with the `-l` parameter (lowercase *L*), which forces `grep` to print the name of each file that contains at least one match without printing the matching text. If a file contains more than one match, it is still printed only once. Alternatively, the `-c` parameter prints each filename that was searched and includes the number of matches at the end, even if there were no matches.

You have a lot of control when specifying the pattern to search for. You can, as shown previously, specify a simple string like `some text`, or you can invert that search by specifying the `-v` parameter. For example, the following returns all the lines of the file `myfile.txt` that do not contain the word `hello`:

```
matthew@seymour:~$ grep -v "hello" myfile.txt
```

You can also use regular expressions for search terms. For example, you can search `myfile.txt` for all references to `cat`, `sat`, or `mat` with this command:

```
matthew@seymour:~$ grep "[cms]at" myfile.txt
```

Adding the `-i` parameter to this removes case-sensitivity, matching `Cat`, `CAT`, `MaT`, and so on:

```
matthew@seymour:~$ grep -i [cms]at myfile.txt
```

You can also control the output to some extent with the `-n` and `--color` parameters. The first tells `grep` to print the line number for each match, which is where it appears in the source file. The `--color` parameter tells `grep` to color the search terms in the output, which helps them stand out from among all the other text on the line. You choose which color you want by using the `GREP_COLOR` environment variable; for example, `export GREP_COLOR=36` gives you cyan, and `export GREP_COLOR=32` gives you lime green.

This next example shows how to use these two parameters to number and color all matches to the previous command:

```
matthew@seymour:~$ grep -in --color [cms]at myfile.txt
```

Later you learn how important `grep` is for piping with other commands.

Paging Through Output with `less`

The `less` command enables you to view large amounts of text in a more convenient way than by using the `cat` command. For example, your `/etc/passwd` file is probably more than a screen long, so if you run `cat /etc/passwd`, you are not able to see the lines at the

top. Using `less /etc/passwd` enables you to use the cursor keys to scroll up and down the output freely. Type `q` to quit and return to the shell.

On the surface, `less` sounds like an easy command; however, it has the infamy of being one of the few Linux commands to have a parameter for every letter of the alphabet. That is, `-a` does something, `-b` does something else, `-c`, `-d`, `-e`,...`-x`, `-y`, `-z`; they all do things, with some letters even differentiating between upper- and lowercase. Furthermore, these parameters are only used when invoking `less`. After you are viewing your text, even more commands are available. Make no mistake; `less` is a complex beast to master.

Input to `less` can be divided into two categories: what you type before running `less` and what you type while running it. The former category is easy, so we start there.

We have already discussed how many parameters `less` can take, but they can be distilled down to three that are very useful: `-M`, `-N`, and `+`. Adding `-M` (which is different from `-m`) enables verbose prompting in `less`. Instead of just printing a colon and a flashing cursor, `less` prints the filename, the line numbers being shown, the total number of lines, and the how far you are through the file, as a percentage. Adding `-N` (which is different from `-n`) enables line numbering.

The last option, `+`, enables you to pass a command to `less` for it to execute as it starts. To use it, you first need to know the commands available to you in `less`, which means it's time to move to the second category of `less` input: what you type while `less` is running.

The basic navigation keys are the up, down, left, and right arrows; Home and End (for navigating to the start and end of a file); and Page Up and Page Down. Beyond that, the most common command is `/`, which initiates a text search. You type what you want to search for and press Enter to have `less` find the first match and highlight all subsequent matches. Type `/` again and press Enter to have `less` jump to the next match. The inverse of `/` is `?`, which searches backward for text. Type `?`, enter a search string, and press Enter to go to the first previous match of that string, or just use `?` and press Enter to go to the next match preceding the current position. You can use `/` and `?` interchangeably by searching for something with `/` and then using `?` to go backward in the same search.

Searching with `/` and `?` is commonly used with the `+` command-line parameter from earlier, which passes `less` a command for execution after the file has loaded. For example, you can tell `less` to load a file and place the cursor at the first match for the search `hello`, like this:

```
matthew@seymour:~$ less +/hello myfile.txt
```

Or, you can use this to place the cursor at the last match for the search `hello`:

```
matthew@seymour:~$ less +?hello myfile.txt
```

Beyond the cursor keys, the controls primarily involve typing a number and then pressing a key. For example, to go to line 50, you type `50g`, or to go to the 75 percent point of the file, you type `75p`. You can also place invisible mark points through the file by pressing `m` and then typing a single letter. Later, while in the same `less` session, you can press

' (a single quote) and then type the letter, and it moves you back to that letter's position. You can set up to 52 marks, named a–z and A–Z.

One clever feature of `less` is that you can, at any time, press `v` to have your file open inside your text editor. This defaults to `vim`, but you can change it by setting the `EDITOR` environment variable to something else.

If you have made it this far, you can already use `less` better than most users. You can, at this point, justifiably skip to the next section and consider yourself proficient with `less`. However, if you want to be a `less` guru, there are two more things to learn: how to view multiple files simultaneously and how to run shell commands.

Like most other file-based commands in Linux, `less` can take several files as its parameters, as in this example:

```
matthew@seymour:~$ less -MN 1.txt 2.txt 3.txt
```

This loads all three files into `less`, starting at `1.txt`. When viewing several files, `less` usually tells you which file you are in and numbers the files: `1.txt (file 1 of 3)` should be at the bottom of the screen. However, certain things make that go away, so you should use `-M` anyway.

You can navigate between files by typing a colon and then pressing `n` to go to the next file or pressing `p` to go to the previous file; these are referred to from now on as `:n` and `:p`. You can open another file for viewing by typing `:e` and providing a filename. This can be any file you have permission to read, including files outside the local directory. Use `Tab` to complete filenames. Files you open in this way are inserted one place after your current position, so if you are viewing file 1 of 4 and open a new file, the new file is numbered 2 of 5 and is opened for viewing right away. To close a file and remove it from the list, use `:d`.

Viewing multiple files simultaneously has implications for searching. By default, `less` searches within only one file, but it is easy to search within all files. When you type `/` or `?` to search, follow it with a `*`. You should see `EOF-ignore` followed by a search prompt. You can now type a search, and it runs through the current file; if nothing is found, the search looks in subsequent files until it finds a match. You can repeat searches by pressing `Esc` and then either `n` or `N`. The lowercase option repeats the search forward across files, and the uppercase option repeats it backward.

The last thing you need to know is that you can get to a shell from `less` and execute commands. The simplest way to do this is just to type `!` and press `Enter`. This launches a shell and leaves you free to type all the commands you want, as per normal. Type `exit` to return to `less`. You can also type specific commands by entering them after the exclamation point (`!`), using the special character `%` for the current filename. For example, `du -h %` prints the size of the current file. Finally, you can use `!!` to repeat the previous command.

Creating Links Between Files with `ln`

Linux allows you to create links between files that look and work like normal files for the most part. Moreover, it allows you to make two types of links, known as hard links and

symbolic links (symlinks). The difference between the two is crucial, although it might not be obvious at first.

Each filename on your system points to what is known as an inode, which is the absolute location of a file. Linux allows you to point more than one filename to a given inode, and the result is a hard link—two filenames pointing to the same file. These two files share the same contents and attributes. So, if you edit one, the other changes because they are both the same file.

On the other hand, a symlink—sometimes called a soft link—is a redirect to the real file. When a program tries to read from a symlink, it automatically is redirected to what the symlink is pointing at. The fact that symlinks are really just dumb pointers has two advantages: You can link to something that does not exist (and create it later if you want), and you can link to directories.

Both types of links have uses. Instead of using redirection, a hard link instead creates a second reference to (name for) the same file, which means that it saves a little bit of time when accessing the file using the symlink. The big difference, however, is that you are able to delete the “original” filename used to create a hard link and the symlinks will still reference the original data file; if you delete the original filename for a soft link, it actually does delete the original data and so all the soft symlinks will break. So why have soft links at all? The reason is that not all file systems permit the use of hard links, and sometimes that matters; for example, when hierarchical data needs to be backed up from a file system that does permit hard links to a file system that does not. In addition, only the root user or a user with `sudo` privileges can create a hard link, whereas soft links are permitted to be created by regular users. Symlinks are popular because they allow a file to appear to be in a different location; you could store your website in `/var/www/live` and an under-construction holding page in `/var/www/construction`. Then you could have Apache point to a symlink `/var/www/html` that is redirected to either the live directory or the construction directory, depending on what you need.

TIP

The `shred` command overwrites a file’s contents with random data, allowing for safe deletion. Because this directly affects a file’s contents, rather than just a filename, this means that all filenames hard linked to an inode are affected.

Both types of link are created using the `ln` command. By default, it creates hard links, but you can create symlinks by passing it the `-s` parameter. The syntax is `ln [-s] something somewhere`, as shown in this example:

```
matthew@seymour:~$ ln -s myfile.txt mylink
```

This creates the symlink `mylink` that points to `myfile.txt`. You don’t see it here, but the file created here is 341 bytes. This is important later. Remove the `-s` to create a hard link. You can verify that your link has been created by running `ls -l`. Your symlink should look something like this:

```
lrwxrwxrwx 1 matthew matthew 5 Feb 19 12:39 mylink -> myfile.txt
```

Note how the file properties start with 1 (lowercase *L*) for *link* and how `ls -l` also prints where a link is going. Symlinks are always very small in size; the previous link is 5 bytes. If you created a hard link, it should look like this:

```
-rw-rw-r 2 matthew matthew 341 Feb 19 12:39 mylink
```

This time the file has normal attributes, but the second number is 2 rather than 1. That number is how many hard links point to this file, which is why it is 2 now. The file size is also the same as that of the previous filename because it is the file, as opposed to just being a pointer.

Symlinks are used extensively in Linux. Programs that have been superseded, such as `sh`, now point to their replacements (in this case `bash`). Sometimes versioning is accomplished through symlinks. For example, applications that link against a fictional program called `snowflake` could load `/usr/bin/snowflake`. Internally, however, that is just a symlink that points to the actual `snowflake` release: `/usr/bin/snowflake.7.2.5`. This enables multiple versions of libraries to be installed without application developers needing to worry about the specific details.

Finding Files from an Index with `locate`

When you use the `find` command, it searches recursively through each directory each time you request a file. As you can imagine, this is slow. Fortunately, Ubuntu ships with a `cron` job that creates an index of all the files on your system every night. Searching this index is extremely fast, which means that if the file you are looking for has been around since the last index, this is the preferable way of searching.

To look for a file in your index, use the command `locate` followed by the names of the files you want to find, like this:

```
matthew@seymour:~$ locate myfile.txt
```

On a relatively modern computer (say, with at least one processor that runs at 1.5GHz or higher), `locate` should be able to return all the matching files in less than one second. The trade-off for this speed is lack of flexibility. You can search for matching filenames, but, unlike with `find`, you cannot search for sizes, owners, access permissions, or other attributes. The one thing you can change is case sensitivity; use the `-i` parameter to do a search that is not case sensitive.

Although Ubuntu rebuilds the filename index nightly, you can force a rebuild whenever you want by running the command `updatedb` with `sudo`. This usually takes a few minutes, but when it's done, the new database is immediately available.

Listing Files in the Current Directory with `ls`

The `ls` command, like `ln`, is a command that most people expect to be very straightforward. It lists files, but how many options can it possibly have? In true Linux style, the answer is many, although again you need only know a few to wield great power!

The basic usage is simply `ls`, which lists the files and directories in the current location. You can filter that by using normal wildcards, so all these are valid:

```
matthew@seymour:~$ ls *
matthew@seymour:~$ ls *.txt
matthew@seymour:~$ ls my*ls *.txt *.xml
```

Any directories that match these filters are recursed into one level. That is, if you run `ls my*` and you have the files `myfile1.txt` and `myfile2.txt` and a directory `mystuff`, the matching files are printed first. Then `ls` prints the contents of the `mystuff` directory.

The most popular parameters for customizing the output of `ls` are as follows:

- ▶ `-a`—Includes hidden files
- ▶ `-h`—Uses human-readable sizes
- ▶ `-l` (*lowercase L*)—Enables long listing
- ▶ `-r`—Reverse order
- ▶ `-R`—Recursively lists directories
- ▶ `-s`—Shows sizes
- ▶ `--sort`—Sorts the listing

All files that start with a period are hidden in Linux, so that includes the `.gnome` directory in your `/home` directory, as well as `.bash_history` and the `.` and `..` implicit directories that signify the current directory and the parent. By default, `ls` does not show these files, but if you run `ls -a`, they are shown. You can also use `ls -A` to show all the hidden files except `.` and `..`.

The `-h` parameter needs to be combined with the `-s` parameter, like this:

```
matthew@seymour:~$ ls -sh *.txt
```

This outputs the size of each matching file in a human-readable format, such as 108KB or 4.5MB.

Using the `-l` parameter enables you to get much more information about your files. Instead of just getting the names of the files, you get output like this:

```
drwxrwxr-x 24 matthew matthew 4096 Dec 24 21:33 arch
-rw-r--r-- 1 matthew matthew 18691 Dec 24 21:34 COPYING
-rw-r--r-- 1 matthew matthew 88167 Dec 24 21:35 CREDITS
drwxrwxr-x 2 matthew matthew 4096 Dec 24 21:35 crypto
```

This output shows four matches and prints a lot of information about each of them. The first row shows the `arch` directory; you can tell it is a directory because its file attributes start with a `d`. The `rw-rwxr-x` following that shows the access permissions, and this has

special meanings because it is a directory. Read access for a directory enables users to see the directory contents, write access enables you to create files and subdirectories, and execute access enables you to `cd` into the directory. If a user has execute access but not read access, the user can `cd` into the directory but cannot list files.

Moving on, the next number on the line is 24, which also has a special meaning for directories: It is the number of subdirectories (including `.` and `..`). After that is `matthew matthew`, which is the name of the user owner and the group owner for the directory. Next are the size and modification time, and finally the directory name itself.

The next line shows the file `COPYING`, and most of the numbers have the same meaning, with the exception of the 1 immediately after the access permissions. For directories, this is the number of subdirectories, but for files it is the number of hard links to this file. A 1 in this column means this is the only filename pointing to this inode, so if you delete it, it is gone.

Ubuntu comes configured with a shortcut command for `ls -l: ll`.

The `--sort` parameter enables you to reorder the output from the default alphabetic sorting. You can sort by various things, although the most popular are extension (alphabetically), size (largest first), and time (newest first). To flip the sorting (making size sort by smallest first), use the `-r` parameter also. So, the following command lists all `.ogg` files, sorted smallest to largest:

```
matthew@seymour:~$ ls --sort size -r *.ogg
```

Finally, the `-R` parameter recurses through subdirectories. For example, `ls /etc` lists all the files and subdirectories in `/etc`, but `ls -R /etc` lists all the files and subdirectories in `/etc`, all the files and subdirectories in `/etc/acpi`, all the files and subdirectories in `/etc/acpi/actions`, and so on until every subdirectory has been listed.

Listing System Information with `lsblk`, `lshw`, `lsmod`, `lspci`, and `neofetch`

The commands `lsblk`, `lshw`, `lsmod`, `lspci`, and `neofetch` are not really related to `ls`, but they work in a similar way. Here, the focus is on listing information about your system rather than the contents of a directory.

To list the storage, or *block*, devices that are attached to your system, use the following:

```
matthew@seymour:~$ lsblk
NAME        MAJ:MIN RM   SIZE RO TYPE MOUNTPOINT
sda         8:0    0 465.8G  0 disk
├─sda1     8:1    0     1K  0 part
├─sda2     8:2    0 453.7G  0 part /
└─sda5     8:5    0  12.1G  0 part [SWAP]
sdb         8:16   0   1.4T  0 disk
└─sdb1     8:17   0   1.4T  0 part
sr0        11:0    1  1024M  0 rom
```

The next command must be run as root for a full listing. Note that the output may be quite long, so this command may be most useful if you pipe it into `grep` and search for a specific bit of text, as described in Chapter 12. To list the hardware detected in your system, use the following:

```
matthew@seymour:~$ sudo lshw
```

To list the status of modules in the Linux kernel, use this, which takes the contents of `/proc/modules` and formats it nicely:

```
matthew@seymour:~$ lsmod
```

To list the PCI devices attached to your system, use the following:

```
matthew@seymour:~$ lspci
```

For an interesting, high-level listing of system information, use this, which will return output similar to Figure 11.1:

```
matthew@seymour:~$ neofetch
```

```

matthew@UUtesting:~$ neofetch
      .-/+00SSSS00+/-..
      `:+SSSSSSSSSSSSSSSS+:`
      ++SSSSSSSSSSSSSSSSyySSSS+-
      .oSSSSSSSSSSSSSSSSdMMMMNYSSSSo.
      /SSSSSSSSSShdmmNNmnyNMMMMHSSSSS/
      +SSSSSSSSShnydMMMMMMNdddySSSSSSSS+
      /SSSSSSShNMMMyhhyyyhNMMMMHSSSSSSS/
      .SSSSSSdMMMMhSSSSSSShNMMMdSSSSSSS.
      +SSShhhyNMMNySSSSSSSSSYNMMMySSSSSSS+
      oSSyNMMMyMMhSSSSSSSSShmmhSSSSSSSo
      oSSyNMMMyMMhSSSSSSSSShmmhSSSSSSSo
      +SSShhhyNMMNySSSSSSSSSYNMMMySSSSSSS+
      .SSSSSSdMMMNhSSSSSSShNMMMdSSSSSSS.
      /SSSSSShNMMMyhhyyyhNMMMMHSSSSSSS/
      +SSSSSSsdnydMMMMMMNdddySSSSSSSS+
      /SSSSSSSSSShdmmNNmnyNMMMMHSSSSS/
      .oSSSSSSSSSSSSSSdMMMMNYSSSSo.
      -+SSSSSSSSSSSSSSyySSSS+-
      `:+SSSSSSSSSSSSSS+:`
      .-/+00SSSS00+/-..
  
```

```

matthew@UUtesting
-----
OS: Ubuntu Focal Fossa (development)
Host: VirtualBox 1.2
Kernel: 5.4.0-21-generic
Uptime: 55 mins
Packages: 3041 (dpkg), 8 (snap)
Shell: bash 5.0.16
Resolution: 1280x800
DE: GNOME
WM: Mutter
WM Theme: Adwaita
Theme: Yaru [GTK2/3]
Icons: Yaru [GTK2/3]
Terminal: gnome-terminal
CPU: Intel Xeon X5690 (8) @ 3.463GHz
GPU: 00:02.0 VMware SVGA II Adapter
Memory: 1248MiB / 15081MiB
  
```

FIGURE 11.1 Neofetch screenshots are a popular way to share info about your system with online friends in places like Reddit.

Reading Manual Pages with `man`

It's time for a much-needed mental break, and the `man` command is easy to use. Most people use only the topic argument, like this: `man gcc`. However, two other commands that work closely with `man` are very useful: `whatis` and `apropos`.

The `whatis` command returns a one-line description of another command, which is the same text you get at the top of that command's man page. Here is an example:

```
matthew@seymour:~$ whatis ls
ls      (1) - list directory contents
```

The output explains what `ls` does and also provides the man page section number for the command so you can query it further.

The `apropos` command takes a search string as its parameter and returns all man pages that match the search. For example, `apropos mixer` returns this list:

```
alsamixer (1) - soundcard mixer for ALSA soundcard driver
mixer (1) - command-line mixer for ALSA soundcard driver
aumix (1) - adjust audio mixer
```

So, you can use `apropos` to help find commands and use `whatis` to find out what a command does.

One neat trick is that many of the tips and tricks you learned for `less` also work when viewing man pages (for example, using `/` and `?` to search). This is one of the beautiful things about UNIX systems: Gaining knowledge in one area often benefits you in other areas.

You can use a number of other commands to search the file system, including the following:

- ▶ **whereis *command***—Returns the location of `command` (for example, `/bin`, `/sbin`, or `/usr/bin/command`) and its man page, which is an entry for the command included as a file already on your computer in a standardized manual format.
- ▶ **whatis *command***—Returns a one-line synopsis from `command`'s man page.
- ▶ **type *name***—Returns how a name would be interpreted if used as a command. This generally shows options or the location of the binary that will be used. For example, `type ls` returns `ls is aliased to 'ls -color=auto'`.

Making Directories with `mkdir`

Making directories is as easy as it sounds, although there is one parameter you should be aware of: `-p`. If you are in `/home/matthew` and you want to create the directory `/home/matthew/audio/sound`, you get an error like this:

```
mkdir: cannot create directory 'audio/sound': No such file or directory
```

At first glance, this seems wrong; `mkdir` cannot create the directory because it does not exist. What it actually means is that it cannot create the directory `sound` because the directory `audio` does not exist. This is where the `-p` parameter comes in: If you try to make a directory within another directory that does not exist, as in the preceding example, `-p` creates the parent directories, too, if you use it like this:

```
matthew@seymour:~$ mkdir -p audio/sound
```

This first creates the `audio` directory and then creates the `sound` directory inside it.

Moving Files with `mv`

The `mv` command is one of the easiest around. There are two helpful parameters to `mv`: `-f`, which overwrites files without asking; and `-u`, which moves the source file only if it is newer than the destination file. That's it. You can use absolute paths to indicate the destination directory (starting from `/`):

```
matthew@seymour:~$ mv filename /newdirectory/newfilename
```

Or you can use relative paths from the current directory (starting without a slash). This is generally entered in the source directory, but it doesn't have to be; you can use an absolute path to indicate the source directory, too.

Renaming Files with `rename`

We often use `mv` to rename a single file. This would be tedious, however, for renaming multiple files. For this, you use `rename`. The basic syntax is simple:

```
matthew@seymour:~$ rename 's/filename/newfilename/'
```

The part in the single quotes is a Perl expression, which means the command is far more powerful than this example suggests. Let's say you have a directory filled with files that end with the extension `.htm`, and you want to rename them all to `.html`:

```
matthew@seymour:~$ rename 's/\.htm /\.html/' *.htm
```

Notice here that the `.` characters must be preceded with a `\` to let the command know they are part of the text being searched and replaced rather than part of the command. This is called “escaping” the characters. Next, notice that the replace part of the command is followed by a wildcard character (`*`) and the remainder of the filename to search for in the directory. Anything matching the combination of `filename.htm` will be renamed to `filename.html`.

`rename` is incredibly powerful. See the man page for more.

Deleting Files and Directories with `rm`

The `rm` command has only one parameter of interest: `--preserve-root`. You should know that issuing `rm -rf /` with `sudo` will try to delete your entire Linux installation because `-r` means recursive and `-f` means force (that is, do not prompt for confirmation before deleting). It used to be possible for a clumsy person to issue this command by accident—not by typing the command on purpose but by putting a space in the wrong place; for example:

```
matthew@seymour:~$ rm -rf /home/matthew
```

This command deletes the `/home` directory of the user `matthew`. This is not an uncommon command; after you have removed a user and backed up the user's data, you will probably want to issue something similar. However, if you accidentally add a space between the `/` and the `h` in `home`, you get this:

```
matthew@seymour:~$ rm -rf / home/matthew
```


This time, the command means “delete everything recursively from / and then delete home/matthew”—quite a different result! To stop this from happening, the `rm` command was changed to use `--preserve-root` by default, meaning that it is impossible to do this accidentally, and `root` now protects you from catastrophe with this message:

```
rm: it is dangerous to operate recursively on '/'
rm: use --no-preserve-root to override this failsafe.
```

Because `--preserve-root` is the default for the command, if you actually do want to remove the contents of your entire file system (this is dangerous), you use `--no-preserve-root`.

Sorting the Contents of a File with `sort`

Say that you have a text file, and you want to sort its contents alphabetically. That is easy. Assume that your text file is filled with one letter on each line, upper- or lowercase. Here is what you get when you run the `sort` command on this file:

```
matthew@seymour:~$ sort testfile.txt
a
A
b
B
```

This is useful. You can also sort in reverse order:

```
matthew@seymour:~$ sort -r testfile.txt
```

This command outputs the same contents but this time from Z to a.

You can use `sort` with files of numbers, but it doesn’t give you a numerical sort. It gives you output like this:

```
matthew@seymour:~$ sort numberfile.txt
1
14
5
58
6
```

You can see here that the `sort` command performs a rather useless alphabetic-style sort on the numbers. Fortunately, there is a switch you can use to fix this:

```
matthew@seymour:~$ sort -n numberfile.txt
1
5
6
14
58
```

There are tons of neat tricks you can do with `sort` and many more parameters explained in the `man` file. For files with a number of columns, such as a directory listing, you can specify the `-k` switch and the number of the column that you want to sort. Let's look at an example of a directory of old `conky` files and start with an `ls` command for comparison:

```
matthew@seymour:~/conky$ ls -la
total 60
drwxr-xr-x  2 matthew matthew 4096 Dec 25  2012 .
drwxr-xr-x 91 matthew matthew 4096 Jul 28 18:42 ..
-rwxr-xr-x  1 matthew matthew 5526 Dec 25  2012 conkyrc_main
-rwxr-xr-x  1 matthew matthew 5502 Dec 25  2012 conkyrc_main~
-rwxr-xr-x  1 matthew matthew 5387 Apr 16  2008 conkyrc_main (old)
-rwxr-xr-x  1 matthew matthew 1326 Mar 15  2008 conkyrc_weather
-rwxr-xr-x  1 matthew matthew 2549 Oct 23  2009 sample_conky.conf
-rwxr-xr-x  1 matthew matthew 128 Apr  8  2008 start_conky (copy).sh
-rwxr-xr-x  1 matthew matthew 139 Dec 25  2012 start_conky.sh
-rwxr-xr-x  1 matthew matthew 140 Dec 25  2012 start_conky.sh~
-rwxr-xr-x  1 matthew matthew 1503 Sep 30  2007 weather.sh
-rwxr-xr-x  1 matthew matthew 2379 Sep 30  2007 weather.xslt
```

Here is the same listing, this time sorted by file size:

```
matthew@seymour:~$ ls -la | sort -n -k5
total 60
-rwxr-xr-x  1 matthew matthew 128 Apr  8  2008 start_conky (copy).sh
-rwxr-xr-x  1 matthew matthew 139 Dec 25  2012 start_conky.sh
-rwxr-xr-x  1 matthew matthew 140 Dec 25  2012 start_conky.sh~
-rwxr-xr-x  1 matthew matthew 1326 Mar 15  2008 conkyrc_weather
-rwxr-xr-x  1 matthew matthew 1503 Sep 30  2007 weather.sh
-rwxr-xr-x  1 matthew matthew 2379 Sep 30  2007 weather.xslt
-rwxr-xr-x  1 matthew matthew 2549 Oct 23  2009 sample_conky.conf
drwxr-xr-x  2 matthew matthew 4096 Dec 25  2012 .
drwxr-xr-x 91 matthew matthew 4096 Jul 28 18:42 ..
-rwxr-xr-x  1 matthew matthew 5387 Apr 16  2008 conkyrc_main (old)
-rwxr-xr-x  1 matthew matthew 5502 Dec 25  2012 conkyrc_main~
-rwxr-xr-x  1 matthew matthew 5526 Dec 25  2012 conkyrc_main
```

Printing the Last Lines of a File with `tail`

If you want to watch a log file as it is written to, or if you want to monitor a user's actions as they are occurring, you need to be able to track log files as they change. In these situations, you need the `tail` command, which prints the last few lines of a file and updates as new lines are added. The following command tells `tail` to print the last few lines of `/var/log/apache2/access.log`, the Apache hit log:

```
matthew@seymour:~$ tail /var/log/apache2/access.log
```

To get `tail` to remain running and update as the file changes, add the `-f` (follow) parameter:

```
matthew@seymour:~$ tail -f /var/log/apache2/access.log
```

You can tie the life span of a `tail -f` to the existence of a process by specifying the `--pid` parameter. When you do this, `tail` continues to follow the file you asked for until it sees that the process identified by *process ID (PID)* is no longer running, at which point it stops tailing.

If you specify multiple files on the command line, `tail` follows them all, printing file headers whenever the input source changes. Press `Ctrl+C` to terminate `tail` when in follow mode.

Printing the Location of a Command with `which`

The purpose of `which` is to tell you exactly which command would be executed if you typed it. For example, `which mkdir` returns `/bin/mkdir`, telling you that running the command `mkdir` runs `/bin/mkdir`.

Downloading Files with `wget`

Let's say you see a website with useful content that you need to download to your server—(for example, <https://releases.ubuntu.com>) because you want to make available to students who work in your computer lab a copy of the ISO of the current release of Ubuntu. This practice, called *mirroring*, is commonly accepted. Or, maybe you want to download the latest *Minecraft* server file to your server and don't want to download it from the website to your local machine first before you upload it to your server. You can do this with `wget`, which exists to download files using HTTP, HTTPS, and FTP, like this:

```
matthew@seymour:~$ wget http://releases.ubuntu.com/20.04/ubuntu-20.04-desktop-amd64.iso
```

This downloads the linked file directly to the directory in which you issue the command.

What if you want to copy all the content files from your existing web server onto a new server? You can use the `-m` or the `--mirror` flag to do this. This example downloads all the contents of the directory you specify, assuming that you have access, to your current directory:

```
matthew@seymour:~$ wget http://youroldserver.com/website/files
```

TIP

You can use `wget` with any standard URL syntax, including specifying ports and usernames and passwords, but you should realize that the username and password information will be transmitted in plain text; therefore, this is not considered a secure method of data transfer. For that, use `sftp`, which is covered in Chapter 19, "Remote Access with SSH and VNC."

References

- ▶ **www.gnu.org**—The website of the GNU project, which contains manuals and downloads for lots of command-line software
- ▶ ***Linux in a Nutshell* by Stephen Figgins, Arnold Robbins, Ellen Siever, and Robert Love**—A dictionary of Linux commands that includes a wide selection of Linux commands and explanations of what they do
- ▶ **www.linuxcommand.org**—A “one-stop command-line shop” that contains a wealth of useful information about the console
- ▶ ***The Art of UNIX Programming* by Eric Raymond**—Focuses on the philosophy behind UNIX and manages to mix in much about the command line

This page intentionally left blank

CHAPTER 12

Command-Line Master Class, Part 2

In Chapter 11, “Command-Line Master Class, Part 1,” you learned a number of useful commands. This chapter follows up with information about how to link commands together to create new command groups. It also looks at the three most popular Linux text editors: `vim`, `emacs`, and `nano`, as well as the `sed` and `awk` tools. This chapter also covers more commands and command-line interface (CLI) tools to help you become successful and efficient. Let’s jump right in.

Redirecting Output and Input

Sometimes, the output of a command is too long to view on one screen. At other times, there might be an advantage to preserving the output as a file, perhaps to be edited later. You can’t do that using `cat` or `less`, at least not using them as described so far. The good news is that it is possible using redirection.

Commands take their input and give their output in a standard place. This is called *standard input* and *standard output*. By default, this is configured as the output of a keyboard for standard input because it comes in to the computer from the keyboard and the screen for standard output because the computer displays the results of a command to the user using that screen. Standard input and standard output can be redirected.

To redirect output, you use `>` on the command line. Sometimes people read this as “in to.” Here is an example:

```
matthew@seymour:~$ cat /proc/cpuinfo > file.txt
```

IN THIS CHAPTER

- ▶ Redirecting Output and Input
- ▶ `stdin`, `stdout`, `stderr`, and Redirection
- ▶ Comparing Files
- ▶ Limiting Resource Use and Job Control
- ▶ Combining Commands
- ▶ Executing Jobs in Parallel
- ▶ Using Environment Variables
- ▶ Using Common Text Editors
- ▶ Working with Compressed Files
- ▶ Using Multiple Terminals with `byobu`
- ▶ Doing a Polite System Reset using `REISUB`
- ▶ Fixing an Ubuntu System that Will Not Boot
- ▶ Tips and Tricks
- ▶ References

You know the first part of this line reads the information about the CPU from the file `/proc/cpuinfo`. Usually this would print it to the screen, but here the output is redirected into a file called `file.txt`, created in your `/home` directory, because that is the directory in which you issued the command. You can now read, edit, or use this file in any way you like.

CAUTION

Be aware that you can overwrite the contents of a file by using a redirect in this manner, so be certain that either the destination file you name does not exist or that its current contents do not need to be preserved.

What if you want to take the contents of an existing file and use that data as an input to a command? It is as simple as reversing the symbol from `>` to `<`:

```
matthew@seymour:~$ cat < file.txt
```

This displays the contents of `file.txt` on the screen. At first glance, that does not seem useful because the command is doing the same thing it usually does: It is printing the contents of a file to the screen. Perhaps a different example would be helpful.

Ubuntu uses a software packaging system called `apt`, which is discussed in Chapter 9, “Managing Software.” By using a command from the `apt` stable, `dpkg`, you can quickly list all software that has been installed using `apt` on a system and record that info into a file by using a redirect:

```
matthew@seymour:~$ sudo dpkg --get-selections > pkg.list
```

This creates a text file named `pkg.list` that contains the list you want. You can open it with a text editor to confirm this. Then you can use this file as input to `dpkg`, perhaps on another system on which you want exactly the same software to be installed:

```
matthew@seymour:~$ sudo dpkg --set-selections < pkg.list
```

This tells `dpkg` to mark for installation any of the items in the list that are not already installed on the second system. One more quick command (included here for completeness of the example, even though it has nothing to do with redirection), and these will be installed:

```
matthew@seymour:~$ sudo apt-get -u dselect-upgrade
```

Earlier in the chapter you saw an example of using `cat` to display several files simultaneously. This example can be modified slightly to redirect the output into a file, thereby making a new file that includes the contents of the previous two, using the order in which they are listed:

```
matthew@seymour:~$ cat myfile.txt myotherfile.txt > combinedfile.txt
```

If you want to append information to the end of a text file, rather than replace its contents, use two greater-than signs, like this:

```
matthew@seymour:~$ echo "This is a new line being added." >> file.txt
```

If you want to suppress the output that you do not want to keep from a process, so that it does not get sent to standard output or saved, send it instead to a special location called the Null Device, like this, where `verboseprocess` is an example of a process that produces lots of output:

```
matthew@seymour:~$ verboseprocess > /dev/null
```

Add the power of redirection to the information in the next section, and you will begin to understand the potential and the power that a command-line-savvy user has and why so many who learn the command line absolutely love it.

stdin, stdout, stderr, and Redirection

When a program runs, it automatically has three input/output streams opened for it: one for input, one for output, and one for error messages. Typically, these are attached to the user's terminal (so they take input from or give output to the command line), but they can be directed elsewhere, such as to or from a file. These three streams are referred to and abbreviated as shown here, and each of these is assigned a number, shown in the third column:

Stream	Abbreviation	Number
Standard input	<code>stdin</code>	0
Standard output	<code>stdout</code>	1
Standard error, or error stream	<code>stderr</code>	2

In the earlier section “Redirecting Output and Input,” you learned how to redirect input and output without needing to know about `stdin` or `stdout`. In addition, you can redirect where the `stderr` messages are sent and also do some more powerful things.

If you are running a program and want any error messages that appear to be directed into a text file instead of having them printed on a screen that might not be monitored, you can use the following command when starting the program or running a command (substitute `program` with the name of the program or command you are running):

```
matthew@seymour:~$ program 2> error.log
```

Here, any error messages from `program` are added to the end of a file named `error.log` in the current working directory.

If you want to redirect both `stderr` and `stdout` to a file, use the following:

```
matthew@seymour:~$ program &> filename
```


You can do the same thing by using the following:

```
matthew@seymour:~$ program >> filename 2>&1
```

In this case, any output from *program* is added to the end of a file named *filename*.

To redirect `stderr` to `stdout`, so that error messages are printed to the screen instead of another location (such as when a program or command is written in a way that already redirects those error messages to a file), use the following:

```
matthew@seymour:~$ program 2>&1
```

Comparing Files

The two things users most commonly want to know when comparing two files are what in the files is the same and what is different. This is especially useful when comparing current versions of configuration files with backup versions of the same files. There are commands to make these tasks easy. Because looking for differences is more common, we start there.

Finding Differences in Files with `diff`

The `diff` command compares files line by line and outputs any lines that are not identical. For example, this command outputs every line that is different between two files:

```
matthew@seymour:~$ diff file1 file2
```

If `file1` and `file2` are different versions of a configuration file—say the current file and a backup—the output quickly tells you what, if anything, has changed. This can help when a config file is automatically updated during an operating system upgrade or when you make a change that doesn't work as well as you had planned and then go back a couple weeks later to change the configuration back.

There are several options you may use when running `diff`. (The original UNIX-style versions, like `-i`, and the newer-style versions, like `--ignore-case`, are identical in what they do; it might simply be easier for you to remember one than the other.) Here are a few of the most useful ones to get you started:

- ▶ `-i` **or** `--ignore-case`—Ignores case differences in file contents
- ▶ `-b` **or** `--ignore-space-change`—Ignores changes in the amount of white space
- ▶ `-w` **or** `--ignore-all-space`—Ignores all white space
- ▶ `-q` **or** `--brief`—Outputs only whether the files differ
- ▶ `-l` **or** `--paginate`—Passes the output through `pr` to paginate it

Finding Similarities in Files with `comm`

The `comm` command compares files line by line and outputs any lines that are identical. For example, this command displays output in three columns, where column 1 shows

lines only in `file1`, column 2 shows lines only in `file2`, and column 3 shows every line that is the same between the two files:

```
matthew@seymour:~$ comm file1 file2
```

This is a much more detailed comparison than with `diff`, and the output can be overwhelming when all you want is to find or check for one or two simple changes. However, it can be incredibly useful when you aren't terribly familiar with either file, and you want to see how they compare.

Fewer options are available when running `comm`. These three are the ones you are most likely to be interested in:

- ▶ `-1`—Suppresses the output of column 1
- ▶ `-2`—Suppresses the output of column 2
- ▶ `-3`—Suppresses the output of column 3

Limiting Resource Use and Job Control

Computer systems run many processes at the same time. This is a good thing and allows users to multitask. Some processes require more system resources than others. Occasionally, a resource-intensive process may take up or require so many resources that it slows down the system for other processes. There are ways to deal with this. This section describes a few of the basics. You must have admin privileges to perform any of the actions in this section.

Listing Processes with `ps`

The `ps` command lists processes and gives you an extraordinary amount of control over its operation. A process is any running program or instance of a running program. There can be many copies of the same program running at the same time, and when that happens, each has its own process. Every process has its own address space or designated part of the computer's memory that is reserved just for this process and its needs. A process group is created when any process begins, and it includes that process and any processes started by it.

In the UNIX/Linux world, a process (parent) has the ability to create another process (child) that executes some given code independently. This can be really useful for programs that need a lot of time to finish. For example, if you have a program that needs to calculate some complex equation, search large databases, or delete and clean up a lot of files, you can write it so that it will “spawn” a child process that performs the task, while the parent returns control to the user. In such a case, the user does not have to wait for the task to finish because the child process is running in the background.

It is important to know is that `ps` is typically used with what are known as BSD-style parameters. In the section “Finding Files by Searching with `find`” in Chapter 11, we discussed UNIX-style, GNU-style, and X-style parameters (`-c`, `--dosomething`, and

-dosomething, respectively); BSD-style parameters are different because they use single letters without dashes.

So, the default use of `ps` lists all processes that you are running that are attached to the terminal. However, you can ask it to list all your processes attached to any terminal (or, indeed, no terminal) by adding the `x` parameter: `ps x`. You can ask it to list all processes for all users with the `a` parameter or combine that with `x` to list all processes for all users, attached to a terminal or otherwise: `ps ax`.

However, both of these are timid compared with the almighty `u` option, which enables user-oriented output. In practice, this makes a huge difference because you get important fields such as the username of the owner, how much CPU time and RAM are being used, when the process was started, and more. This outputs a lot of information, so you might want to try adding the `f` parameter, which creates a process forest by using ASCII art to connect parent commands with their children. You can combine all the options so far with this command: `ps faux`. (Yes, with a little imagination, you can spell words with the parameters.)

You can control the order in which the data is returned by using the `--sort` parameter. This takes either `a +` or `a -` (although the `+` is the default) followed by the field you want to sort by: `command`, `%cpu`, `pid`, and `user` are all popular options. If you use the minus sign, the results are reversed. The following command lists all processes, ordered by CPU usage in descending order (with output abridged to conserve space):

```
matthew@seymour:~$ ps aux --sort=-%cpu
USER      PID %CPU %MEM    VSZ   RSS TTY      STAT START   TIME COMMAND
matthew   17669 10.2  3.8  5256448 1603116 ?        Sl   08:30    5:23  /usr/lib/virtua
matthew    6761  4.4  0.0  1226128 17544 ?        SNl  Jan26   951:13  conky -d -c /ho
matthew   30849  3.8  0.7  1425084 310356 ?        Sl   09:20    0:04  /opt/google/chr
matthew   18668  3.2  5.5  3692116 2297988 ?        SLl  Feb05  227:08  /opt/google/chr
matthew   31211  3.0  0.0  698532  39348 ?        Sl   09:22    0:00  /usr/bin/xfce4-
matthew   14553  2.8  1.0  2881828 422216 ?        Sl   08:17    1:50  /usr/lib/libreo
root       1379  2.3  0.7  780372  304860 tty7     Ssl+ Jan26  490:58  /usr/lib/xorg/X
matthew   31215  0.4  0.0  21952   5524 pts/0    Ss   09:22    0:00  bash
systemd+  1316  0.2  0.0  66008   6284 ?        Ss   Jan26   54:00  /lib/systemd/sy
matthew   5369  0.1  0.0  457476  39304 ?        SLl  Jan26   31:23  /usr/bin/gnome-
matthew   5589  0.1  0.0  210212  30692 ?        S    Jan26  22:40  xfwm4 --replace
matthew   5661  0.1  0.0  1840276 13292 ?        S<l  Jan26  26:51  /usr/bin/pulsea
root        1  0.0  0.0  220548  8952 ?        Ss   Jan26    0:15  /sbin/init spla
root        2  0.0  0.0      0      0 ?        S    Jan26    0:00  [kthreadd]
root        4  0.0  0.0      0      0 ?        S<   Jan26    0:00  [kworker/0:0H]
root        7  0.0  0.0      0      0 ?        S<   Jan26    0:00  [mm _percpu_wq]
root        8  0.0  0.0      0      0 ?        S    Jan26    0:08  [ksoftirqd/0]
root        9  0.0  0.0      0      0 ?        S    Jan26    5:22  [rcu _sched]
root       10  0.0  0.0      0      0 ?        S    Jan26    0:00  [rcu _bh]
root       11  0.0  0.0      0      0 ?        S    Jan26    0:01  [migration/0]
root       12  0.0  0.0      0      0 ?        S    Jan26    0:01  [watchdog/0]
root       13  0.0  0.0      0      0 ?        S    Jan26    0:00  [cpuhp/0]
```

There are many other parameters for `ps`, including a large number of options for compatibility with other UNIXes, all of which can be found in the man page.

Listing Jobs with `jobs`

A *job* is any program you interactively start that doesn't then detach from the user and run on its own (like a daemon does). If you're running an interactive program, you can press `Ctrl+Z` to suspend it. Then you can start it back in the foreground (using `fg`, covered next) or in the background (using `bg`, covered with `fg`).

While the program is suspended or running in the background, you can start another program. You then have two jobs running. You can also start a program running in the background by appending an `&` like this:

```
matthew@seymour:~$ programname &
```

When started this way, a program runs as a background job. To list all the jobs you are running, you can use `jobs`:

```
matthew@seymour:~$ jobs
```

You can use several useful parameters with `jobs`, including the following:

- ▶ `-l`—Lists the process IDs along with the normal output
- ▶ `-n`—Displays information only about jobs that have changed status since the user was last notified of their status
- ▶ `-r`—Restricts output to running jobs
- ▶ `-s`—Restricts output to stopped jobs

Running One or More Tasks in the Background

Put the `&` (ampersand) symbol at the end of any command to make it run in the background:

```
matthew@seymour:~$ command &
[1] 11423
```

A background process runs without any user input. The shell is not forced to wait until the process is complete before it is freed up to allow more commands to be input and run. When you tell a command to run in the background, you are given its job number in brackets followed by its PID, or process ID number. You can use this to manage the process later, if necessary.

You can input a list of several commands to run in the background. Each will run separately and have its own PID. In this sample, `a`, `b`, and `c` represent commands:

```
matthew@seymour:~$ a & b & c &
[1] 11427
```

```
[2] 11428
[3] 11429
```

You can even use pipes within background processes, and you can combine multiples of each. The letters here represent individual commands:

```
matthew@seymour:~$ d & | e & f & g & | h &
[1] 11432
[2] 11433
[3] 11434
```

Notice that the line above becomes three separate background processes, even though five commands were issued. That is because commands that are piped together are treated as one process.

Moving Jobs to the Background or Foreground with `bg` and `fg`

The shell has the concept of foreground jobs and background jobs. *Foreground jobs* are process groups with control of the terminal, and *background jobs* are process groups without control of the terminal.

Let's say you start a job by running a program at the command line—maybe something like this, which could take a while to run:

```
matthew@seymour:~$ find . -type f -printf "%s\t%p\n" | sort -n
```

When you run this, it starts in the foreground, meaning the terminal is interactive with you only for this job, until the job is complete. This particular job will find all files in the current directory and its subdirectories and list them according to their size. You wouldn't likely want to tie up your terminal session the whole time the job is running. Say that you mean to run it with an `&` at the end so that it will run in the background, but you forget. No worries. You can press `Ctrl+Z` to suspend the job, and then you type this:

```
matthew@seymour:~$ bg
```

That's it. This causes the process to resume, but this time running in the background.

Both `bg` and `fg`, if entered with no further arguments, operate on the job you have most recently interacted with.

Remember that the `jobs` command lists all current jobs and their status (running, stopped, and so on). If you want to move a job running in the background to the foreground, first list the running jobs. Each one has a number next to it in the listing. Use the job number to move a job to the foreground, like this:

```
matthew@seymour:~$ fg %2
```

If you want to move a specific job to the background, just press `Ctrl+Z` and add the job number the same way:

```
matthew@seymour:~$ bg %2
```

Remember that jobs running in this manner terminate when the shell is closed. If you want a job to continue after you exit, you should consider using a tool such as `byobu`, covered later in this chapter, or learn to run the process as a daemon, which is beyond the scope of this chapter and requires you to do some further research.

Printing Resource Usage with `top`

The `top` command is unusual in this list because the few parameters it takes are rarely, if ever, used. Instead, it has a number of commands you can use while it is running to customize the information it shows you. To get the most from these instructions, open two terminal windows. In the first one, run the program `yes` and leave it running; in the second one, run `top`.

When you run `top`, you see a display like the one shown in Figure 12.1.

```
top - 18:10:40 up 48 min, 2 users, load average: 0.73, 0.83, 0.49
Tasks: 135 total, 1 running, 134 sleeping, 0 stopped, 0 zombie
Cpu(s): 0.3%us, 0.3%sy, 0.0%ni, 99.0%id, 0.0%wa, 0.0%hi, 0.3%si, 0.0%st
Mem: 508488k total, 393612k used, 114876k free, 54008k buffers
Swap: 916476k total, 0k used, 916476k free, 180944k cached
```

PID	USER	PR	NI	VIRT	RES	SHR	S	%CPU	%MEM	TIME+	COMMAND
1027	root	20	0	163m	21m	7960	S	0.3	4.4	0:18.07	Xorg
1371	root	20	0	5808	2868	2332	S	0.3	0.6	0:05.74	vmtoolsd
1764	matthew	20	0	46564	20m	16m	S	0.3	4.1	0:13.12	vmware-user-loa
2916	matthew	20	0	61572	13m	10m	S	0.3	2.7	0:00.45	gnome-terminal
2941	matthew	20	0	2624	1116	840	R	0.3	0.2	0:00.20	top
1	root	20	0	2868	1700	1224	S	0.0	0.3	0:01.45	init
2	root	20	0	0	0	0	S	0.0	0.0	0:00.00	kthreadd
3	root	20	0	0	0	0	S	0.0	0.0	0:00.16	ksoftirqd/0
4	root	RT	0	0	0	0	S	0.0	0.0	0:00.00	migration/0
5	root	RT	0	0	0	0	S	0.0	0.0	0:00.00	watchdog/0
6	root	20	0	0	0	0	S	0.0	0.0	0:00.12	events/0
7	root	20	0	0	0	0	S	0.0	0.0	0:00.00	cpuset
8	root	20	0	0	0	0	S	0.0	0.0	0:00.00	khelper
9	root	20	0	0	0	0	S	0.0	0.0	0:00.00	netns
10	root	20	0	0	0	0	S	0.0	0.0	0:00.00	asynmgr

FIGURE 12.1 Use the `top` command to monitor and control processes.

The default sort order in `top` shows the most CPU-intensive tasks first. The first command there should be the `yes` process you just launched from the other terminal, but there should be many others also. Say that you want to filter out all the other users and focus on the user running `yes`. To do this, press `u` and enter the username you used when you ran `yes`. When you press Enter, `top` filters out processes not being run by that user.

The next step is to kill the PID of the `yes` command, so you need to understand what each of the important fields means:

- ▶ **PID**—The process ID
- ▶ **User**—The owner of the process
- ▶ **PR**—Priority
- ▶ **NI**—Niceness

- ▶ **virt**—Virtual image size, in kilobytes
- ▶ **res**—Resident size, in kilobytes
- ▶ **shr**—Shared memory size, in kilobytes
- ▶ **s**—Status
- ▶ **%CPU**—CPU usage
- ▶ **%Mem**—Memory usage
- ▶ **Time+**—CPU time
- ▶ **Command**—The command being run

Several of these fields are unimportant unless you have a specific problem. The important ones in this case are `PID`, `User`, `Niceness`, `%CPU`, `%MEM`, `Time+`, and `Command`. The `Niceness` of a process is how much time the CPU allocates to it compared to everything else on the system; `19` is the lowest, and `-20` is the highest.

With the columns explained, you should be able to find the `PID` of the errant `yes` command launched earlier; it is usually the first number below `PID`. Now type `k`, enter that `PID`, and press Enter. You are prompted for a signal number (the manner in which you want the process killed), with `15` provided as the default. Signal `15` (also known as `SIGTERM`, for *terminate*) is a polite way of asking a process to shut down, and all processes that are not wildly out of control should respond to it. Give `top` a few seconds to update itself, and hopefully the `yes` command should be gone. If not, you need to be more forceful: Type `k` again, enter the `PID`, and press Enter. When prompted for a signal to send, enter `9` and press Enter to send `SIGKILL`, or “terminate whether you like it or not.”

You can choose the fields to display by pressing `f`. A new screen appears, listing all possible fields, along with the letter you need to press to toggle their visibility. Selected fields are marked with an asterisk and have their letter, as follows:

```
* A: PID          = Process Id
```

If you now press the `a` key, the screen changes to this:

```
a: PID          = Process Id
```

When you have made your selections, press Enter to return to the normal `top` view with your normal column selection.

You can also press `F` to select the field you want to use for sorting. This works the same way as the field selection screen, except that you can select only one field at a time. Again, press Enter to get back to `top` after you have made your selection, and it is updated with the new sorting.

If you press `B`, text bolding is enabled. By default, this bolds some of the header bar as well as any programs that are currently running (as opposed to sleeping), but if you press `x`, you can also enable bolding of the sorted column. You can use `y` to toggle bolding of running processes.

The last command to try is `r`, which enables you to renice—or adjust the niceness value of—a process. You need to enter the PID of the process, press Enter, and enter a new niceness value. Remember that 19 is the lowest and -20 is the highest; anything less than 0 is considered “high” and should be used sparingly.

You can combine the information you learn here with the information in Chapter 16, “System-Monitoring Tools,” for even more power over your system.

Setting Process Priority with `nice`

You can set the priority for individual processes to tell the kernel either to limit or give extra priority to a specific process. This is most useful when multiple concurrent processes are demanding more resources than are actually available because this is the condition that generally causes slowdowns and bottlenecks. Processes set with a higher priority get a larger portion of the CPU time than lower-priority processes.

You can set the process priority when you first run a program by putting the command `nice` before whatever you are going to run and assigning the process a value that designates its priority. By default, all processes start with a priority of 0 (zero). `nice` can be set that to a maximum of -20, which is the highest priority, to a minimum of 19, the lowest priority. Only root can increase the priority of a process (set negative nice values) but any user can lower the priority of a process (set positive values).

Here is an example that takes the `tar` command used earlier in this chapter and sets its priority very low, because `tar` can demand significant system resources but is often not something whose output you require immediately. You could run the same command as earlier but with `nice` set to 19 to allow you to do something else with the system at the same time:

```
matthew@seymour:~$ sudo nice -n 19 tar czf compressedfilename.tgz directoryname
```

If a process is already running, you can reset the priority (some say “renice it”) by using `renice`. To adjust a specific process, first use `top` to learn the PID for the process and then use `-p PID`, as shown in this example that renices PID 20136 to priority 19:

```
matthew@seymour:~$ sudo renice 19 -p 20136
```

This command is a little more flexible, as it also allows priority adjustments to be made on all processes owned by a specific user or group in the system. Notice that `renice` is most commonly used to lower the priority of a system-slowng task, but it can also be used to bump up the priority for an urgent task, as shown in these examples. Here you first give all tasks by the user `mysql` (using `-u username`) a priority of -20 (top priority, remember?) and then give all tasks belonging to system users in the `website` group (using `-g group-name`) a priority of -20:

```
matthew@seymour:~$ sudo renice -20 -u mysql
matthew@seymour:~$ sudo renice -20 -g website
```


With the `ionice` command, you can adjust priority for disk access, similar to how `nice` and `renice` set priority for CPU access. The difference here is that there are only three class settings for priority. The lowest priority is Idle (3), which permits the process to access the disk only when other processes are not using the disk. In the middle is Best Effort (2), which is the default and allows the kernel to schedule access as its algorithms deem appropriate. The highest priority is Real Time (1), which gives this process the first access to the disk whenever it demands it, regardless of what other processes are running. The Real Time setting can be dangerous because it can cause other processes to lose their data; this isn't guaranteed to happen, but you should consider yourself warned, and you probably want to spend some time studying the man page for `ionice` before you set anything to Real Time.

To use `ionice`, find the PID for a process and use it to set the priority. Notice that there are no spaces between the flag and the value using this command. Here is an example of setting set the priority (using the `-c` flag, for `class`) to 3 for PID 24351 (using the `-p` flag):

```
matthew@seymour:~$ sudo ionice -c3 -p24351
```

You can find other useful tips for managing your system in Chapter 16.

Combining Commands

So far in this chapter, you have been using commands only individually—and for the most part, that is what you do in practice. However, some of the real power of these commands lies in the capability to join them together to get exactly what you want. There are some extra little commands that we have not looked at that are often used as glue because they do one very simple thing that enables a more complex process to work.

Pipes

All the commands we have looked at have printed their information to the screen, but this is often flexible.

A pipe is a connector between one command's output and another command's input. Instead of sending its output to your terminal, you can use a pipe to send that output directly to another command as input.

Two of the commands we have looked at so far are `ps` and `grep`: the process lister and the string matcher. You can combine the two to find out which users are playing NetHack right now:

```
matthew@seymour:~$ ps aux | grep nethack
```

This creates a list of all the processes running right now and sends that list to the `grep` command, which filters out all lines that do not contain the word `nethack`. Ubuntu allows you to pipe as many commands as you can sanely string together. For example, you could add the `wc` command, which counts the numbers of lines, words, and characters in its input, to count precisely how many times *NetHack* is being run:

```
matthew@seymour:~$ ps aux | grep nethack | wc -l
```

The `-l` (lowercase *L*) parameter to `wc` prints only the line count.

Using pipes in this way is often preferable to using the `-exec` parameter with `find`, simply because many people consider `find` to be a black art, and using it less frequently is better. This is where the `xargs` command comes in: It converts output from one command into arguments for another.

For a good example, consider this mammoth `find` command from earlier:

```
matthew@seymour:~$ find / -name "*.txt" -size +10k -user matthew -not -perm +o-r
[ccc]-exec chmod o+r {} \;
```

This searches every directory from `/` onward for files matching `*.txt` that are greater than 10KB, are owned by user `matthew`, and do not have read permission for others. Then it executes `chmod` on each of the files. It is a complex command, and people who are not familiar with the workings of `find` might have problems understanding it. You can break it into two—a call to `find` and a call to `xargs`. The simplest conversion would look like this:

```
matthew@seymour:~$ find / -name "*.txt" -size +10k -user matthew -not -perm +o-r |
[ccc]xargs chmod o+r
```

This has eliminated the confusing `{ } \;` from the end of the command, but it does the same thing—and faster, too. The speed difference between the two is because using `-exec` with `find` causes it to execute `chmod` once for each file. However, `chmod` accepts many files at a time and, because you are using the same parameter each time, you should take advantage of that. The second command, using `xargs`, is called once with all the output from `find`, and it therefore saves many command calls. The `xargs` command automatically places the input at the end of the line, so the previous command might look something like this:

```
matthew@seymour:~$ xargs chmod o+r file1.txt file2.txt file3.txt
```

Not every command accepts multiple files, though, and if you specify the `-l` parameter, `xargs` executes its command once for each line in its input. If you want to check what it is doing, use the `-p` parameter to have `xargs` prompt you before executing each command.

For even more control, the `-i` parameter allows you to specify exactly where the matching lines should be placed in your command. This is important if you need the lines to appear before the end of the command or need it to appear more than once. Either way, using the `-i` parameter also enables the `-l` parameter so each line is sent to the command individually. The following command finds all files in `/home/matthew` that are larger than 10,000KB (10MB) and copies them to `/home/matthew/archive`:

```
matthew@seymour:~$ find /home/matthew -size +10000k | xargs -i cp {} ./home/
matthew/
[ccc]archive
```

Using `find` with `xargs` is a unique case. All too often, people use pipes when parameters would do the job just as well. For example, the following two commands are identical:

```
matthew@seymour:~$ ps aux --sort=-%cpu | grep -v `whoami`
matthew@seymour:~$ ps -N ux --sort=-%cpu
```

The former prints all users and processes and then pipes that to `grep`, which in turn filters out all lines that contain the output from the program `whoami` (the username in this case). So, the first line prints all processes being run by other users, sorted by CPU use. The second line does not specify the `a` parameter to `ps`, which makes it list only your parameters. It then uses the `-N` parameter to flip that, which means it is everyone but you, without the need for `grep`.

The reason people use the former is often just simplicity: Many people know only a handful of parameters to each command, so they can string together two commands simply rather than write one command properly. Unless the command is to be run regularly, this is not a problem. Indeed, the first line would be better because it does not drive people to the manual pages to find out what `ps -N` does.

You can string together any commands that use standard input and output formats. Another useful example is the following series, which verifies the installation of a named software package (in this example, a search for FTP-related software):

```
matthew@seymour:~$ dpkg --get-selections | grep ftp | sort
ftp                install
lftp                install
```

Here, `dpkg` is being told to list all installed packages, `grep` is searching that list for any line containing `ftp`, and `sort` is sorting alphabetically (which is not vital in this two-line example, but it is really useful if you have a large number of results).

Combining Commands with Boolean Operators

If you want to run a second command only if the first command is successfully completed, you can. Every command you issue to the system outputs an exit status: `0` for true (successful) and `1` for false (failed). The system receives these even if they are not displayed to the user. The `&&` operator, when added to the end of a command, reads that exit status and confirms its value as `0` for true before allowing the next command to be run. Again, the letters represent commands:

```
matthew@seymour:~$ i && k
```

You can do exactly the opposite with `||`, which runs the following command only if the first one returns an exit status of `1` for false:

```
matthew@seymour:~$ m || n
```

Running Separate Commands in Sequence

If you want to have a set of commands run in order but not use the output from one as the input to the next one, you can. Separating commands with a `;` (semicolon) causes the

system to treat each item as if it were entered on a new line after the previous item finished running. Let's say you have three commands: `doctor`, `rose`, and `tardis`. You could run each in order by using this command:

```
matthew@seymour:~$ doctor ; rose ; tardis
```

Note that the spaces before and after the semicolons are optional, but they do make the line easier to read.

Process Substitution

Sometimes the output of one or more commands is precisely what you want to use as the input to another command. You can use output redirection for this purpose, using what we call *process substitution*. In process substitution, you surround one or more commands with `()` and precede each list with a `<`. When you do this, do *not* insert a space between the `<` and the opening `(`. The resulting command looks like this:

```
matthew@seymour:~$ cat <(ls -al)
```

This first example is really the same as `ls -al | cat`. With only the output of one process being involved, it doesn't seem worth learning an additional command.

In the following example, you take the output of two `ls` commands as input to a `diff` command to compare the contents of two directories:

```
matthew@seymour:~$ diff <(ls firstdirectory) <(ls seconddirectory)
```

This is faster because you don't have to wait for temporary files to be written and then read; it saves both disk space and the time needed to clean up temporary files. One especially neat advantage of doing this is that `bash` automatically runs the multiple tasks being used as input in parallel, which is faster than doing them sequentially with redirects, like this:

```
matthew@seymour:~$ ls firstdirectory > file1.txt
matthew@seymour:~$ ls seconddirectory > file2.txt
matthew@seymour:~$ diff file1.txt file2.txt
```

Executing Jobs in Parallel

When you issue a command in the terminal, it is executed by the CPU (the central processing unit, or processor). Common processors include Intel's i3, i5, i7, and Xeon series as well as AMD's Ryzen and Epyc. Most computers today have more than one processor (more accurately stated as "processor core," each of which essentially functions as a separate processor). This allows multiple jobs to run at the same time. Programs that are written to take advantage of multiple processors or cores concurrently can run much faster by splitting large tasks into chunks and sending the chunks to multiple processors or cores in parallel.

Most terminal commands are not (yet?) written to take advantage of multiple processors or cores. So, when you execute a command, generally the command uses whatever

percentage it requires of one core until the task is complete. Doing something that requires significant amounts of processing can take a long time.

GNU `parallel` is a shell tool for executing jobs in parallel across multiple processors, cores, or even multiple connected computers. The jobs can be simple single commands or even scripts, as long as they are executables or functions. The tool is powerful and complex and deserves a book of its own. For more information, see the official documentation at www.gnu.org/software/parallel/ and the official tutorial at www.gnu.org/software/parallel/parallel_tutorial.html.

Using Environment Variables

A number of in-memory variables are assigned and loaded by default when you log in. These variables, known as *environment variables*, can be used by various commands to get information about your environment, such as the type of system you are running, your `/home` directory, and the shell in use. Environment variables are used to help tailor the computing environment of your system and include helpful specifications and setup, such as default locations of executable files and software libraries. If you begin writing shell scripts, you might use environment variables in your scripts. Until then, you need to be aware only of what environment variables are and do.

The following list includes a number of environment variables, along with descriptions of how the shell uses them:

- ▶ **PWD**—Provides the full path to your current working directory, used by the `pwd` command, such as `/home/matthew/Documents`
- ▶ **USER**—Declares the user's name, such as `matthew`
- ▶ **LANG**—Sets the default language, such as `English`
- ▶ **SHELL**—Declares the name and location of the current shell, such as `/bin/bash`
- ▶ **PATH**—Sets the default locations of executable files, such as `/bin`, `/usr/bin`, and so on
- ▶ **TERM**—Sets the type of terminal in use, such as `vt100`, which can be important when using screen-oriented programs, such as text editors

You can print the current value of any environment variable by using `echo $VARIABLE-NAME`, like this:

```
matthew@seymour:~$ echo $USER
matthew
matthew@seymour:~$
```

NOTE

Each shell can have its own feature set and language syntax, as well as a unique set of default environment variables.

You can use the `env` or `printenv` command to display all environment variables, as follows:

```
matthew@seymour:~$ env
ORBIT_SOCKETDIR=/tmp/orbit-matthew
SSH_AGENT_PID=1729
TERM=xterm
SHELL=/bin/bash
WINDOWID=71303173
GNOME_KEYRING_CONTROL=/tmp/keyring-qTEFTw
GTK_MODULES=canberra-gtk-module
USER=matt
hew
SSH_AUTH_SOCK=/tmp/keyring-qTEFTw/ssh
DEFAULTS_PATH=/usr/share/gconf/gnome.default.path
SESSION_MANAGER=local/seymour:/tmp/.ICE-unix/1695
USERNAME=matthew
XDG_CONFIG_DIRS=/etc/xdg/xdg-gnome:/etc/xdg
DESKTOP_SESSION=gnome
PATH=/usr/local/sbin:/usr/local/bin:/usr/sbin:/usr/bin:/sbin:/bin:/usr/games
PWD=/home/matthew
hew
GDM_KEYBOARD_LAYOUT=us
LANG=en_US.utf8
GNOME_KEYRING_PID=1677
MANDATORY_PATH=/usr/share/gconf/gnome.mandatory.path
GDM_LANG=en_US.utf8
GDMSESSION=gnome
HISTCONTROL=ignoreboth
SPEECHD_PORT=7560
SHLVL=1
HOME=/home/matthew
hew
LOGNAME=matthew
hew
LESSOPEN=| /usr/bin/lesspipe %s
DISPLAY=:0.0
LESSCLOSE=/usr/bin/lesspipe %s %s
XAUTHORITY=/var/run/gdm/auth-for-matthew-PzcGqF/database
COLORTERM=gnome-terminal
OLDPWD=/var/lib/mlocate
_=/usr/bin/env
```

This abbreviated list shows some of the common variables, which are set by configuration, or *resource*, files contained in `/etc` and `/etc/skel` and are copied to the user's `/home` directory when it is created. You can find default settings for `bash`, for example, in

`/etc/profile` and `/etc/bashrc` as well as `.bashrc` or `.bash_profile` files in your `/home` directory. Read the man page for `bash` for details about using these configuration files.

One of the most important environment variables is `$PATH`, which defines the location of executable files. For example, if, as a regular user, you try to use a command that is not located in your `$PATH` (such as the imaginary `command` command), you see something like this:

```
matthew@seymour:~$ command
-bash: command: command not found
```

If the command you're trying to execute exists in the Ubuntu software repositories but is not yet installed on your system, Ubuntu responds with the correct command to install the command:

```
matthew@seymour:~$ command
```

The program 'command' is currently not installed. You can install it by typing:

```
sudo apt-get install command
```

However, you might know that the command is definitely installed on your system, and you can verify it by using the `whereis` command, like this:

```
matthew@seymour:~$ whereis command
command: /sbin/command
```

You can also run the command by typing its full pathname or complete directory specification, as follows:

```
matthew@seymour:~$ sudo /sbin/command
```

As you can see in this example, the `command` command is indeed installed. What happened is that by default, the `/sbin` directory is not in your `$PATH`. One of the reasons for this is that commands under the `/sbin` directory are normally intended to be run only by root. You can add `/sbin` to your `$PATH` by editing the file `.bash_profile` in your `/home` directory (if you use the `bash` shell by default, as most Linux users). Look for the following line:

```
PATH=$PATH:$HOME/bin
```

You can then edit this file, perhaps using one of the text editors discussed later in this chapter, to add the `/sbin` directory, like so:

```
PATH=$PATH:/sbin:$HOME/bin
```

Save the file. The next time you log in, the `/sbin` directory is in your `$PATH`. One way to use this change right away is to read in the new settings in `.bash_profile` by using the `bash` shell's `source` command, as follows:

```
matthew@seymour:~$ source .bash_profile
```

You can now run commands located in the `/sbin` directory without the need to explicitly type the full pathname.

Some Linux commands also use environment variables to acquire configuration information (such as a communications program looking for a variable such as `BAUD_RATE`, which might denote a default modem speed).

To experiment with the environment variables, you can modify the `PS1` variable to manipulate the appearance of your shell prompt. If you are working with `bash`, you can use its built-in `export` command to change the shell prompt. For example, say that your default shell prompt looks like this:

```
matthew@seymour:~$
```

You can change its appearance by using the `PS1` variable, like this:

```
matthew@seymour:~$ export PS1='$OSTYPE r001z ->'
```

After you press Enter, you see the following:

```
linux-gnu r001z ->
```

NOTE

See the `bash` man page for other variables you can use for prompt settings.

Using Common Text Editors

Linux distributions include a number of applications known as *text editors* that you can use to create text files or edit system configuration files. Text editors are similar to word processing programs but generally have fewer features, work only with text files, and might or might not support spell checking or formatting. Text editors range in features and ease of use and are found on nearly every Linux distribution. The number of editors installed on a system depends on what software packages you've installed on the system.

The most popular console-based text editors include the following:

- ▶ **emacs**—The comprehensive GNU `emacs` editing environment, which is much more than an editor; see the section “Working with `emacs`,” later in this chapter
- ▶ **nano**—A simple text editor similar to the classic `pico` text editor that was included with the once-common `pine` email program
- ▶ **vim**—An improved compatible version of the `vi` text editor (which we call `vi` in the rest of this chapter because it has a symbolic link named `vi` and a symbolically linked manual page)

Note that not all text editors are *screen oriented*, meaning designed for use from a terminal. The following are some of the text editors designed to run from a graphical desktop and that provides a graphical interface with menu bars, buttons, scrollbars, and so on:

- ▶ **gedit**—A GUI text editor for GNOME, which is installed by default with Ubuntu
- ▶ **kate**—A simple KDE text editor
- ▶ **kedit**—Another simple KDE text editor

A good reason to learn how to use a text-based editor, such as `vi` or `nano`, is that system maintenance and recovery operations almost never take place during GUI sessions, negating the use of a GUI editor. Many larger, more complex and capable editors do not work when Linux is booted to its single-user or maintenance mode. If anything goes wrong with your system, and you can't log in to the GUI, knowledge and experience of using both the command line and text editors will be very important. Make a point of opening some of the editors and playing around with them. You never know; you might just thank us someday.

Another reason to learn how to use a text-based editor under the Linux console mode is so that you can edit text files through remote shell sessions because many servers do not host graphical desktops.

NOTE

Before you take the time to get familiar with a nonstandard text editor, consider this: All three of the editors discussed here are readily available and common. Two of them, `nano` and `vi`, are almost universally installed. If you spend your time learning a nonstandard editor, you will find yourself having to install it on every system or fighting against the software that is already there instead of using your time productively. Feel free to use any text editor you prefer, but we strongly recommend that you make sure you have at least a basic working knowledge of these standard editors so that you can walk up to any system and start working when necessary.

Working with `nano`

We discuss `nano` first because it has the easiest learning curve. It is neither the most powerful nor the most “guru approved,” but `nano` is a respectable text editor that you can run from the command line, and it's often perfect for quick tasks such as editing configuration files.

Learning how to use `nano` is quick and easy. You might need to edit files on a Linux system with a minimal install or a remote server without a more extensive offering of installed text editors. Chances are nearly 100 percent that `nano` will be available.

You can start an editing session by using the `nano` command like this:

```
matthew@seymour:~$ nano file.txt
```

When you first start editing, you see the text on the screen with a title bar across the top and a list of simple commands across the bottom. The editor is simple enough that you can use it without any instruction. Here are the basic commands, just so you can compare them with other editors discussed here:

- ▶ **Cursor movement**—Arrow keys (left, down, up, and right), Page Up and Page Down keys, or Ctrl+Y and Ctrl+V page up and down
- ▶ **Add characters**—Type at the cursor location
- ▶ **Delete character**—Backspace or Delete
- ▶ **Exit**—Ctrl+X (prompts to ask whether to save changes)
- ▶ **Get Help**—Ctrl+G

NOTE

`nano` really is very easy to use, but this does not mean it cannot be used by power users. Take a little time and read the contents of Help to discover some of the interesting and powerful capabilities of this editor.

Working with `vi`

The one editor found on nearly every UNIX and Linux system is the `vi` editor, originally written by Bill Joy. This simple-to-use but incredibly capable editor features a somewhat cryptic command set, but you can put it to use with only a few commands. Although many experienced UNIX and Linux users use `vi` extensively during computing sessions, many users who do only quick and simple editing might not need all its power and might prefer an easier-to-use text editor such as `nano`. Diehard GNU fans and programmers often use `emacs` for pretty much everything.

However, learning how to use `vi` is a good idea. You might need to edit files on a Linux system with a minimal install or on a remote server without a more extensive offering of installed text editors. Chances are nearly 100 percent that `vi` will be available.

You can start an editing session by using the `vi` command like this:

```
matthew@seymour:~$ vi file.txt
```

The `vi` command works by using an insert (or editing) mode and a viewing (or command) mode.

When you first start editing, you are in the viewing mode. You can use your arrow or other navigation keys (as shown later) to scroll through the text. To start editing, press the `i` key to insert text or the `a` key to append text. When you're finished, use the Esc key to toggle out of the insert or append modes and into the viewing (or command) mode. To enter a command, type a colon (:), followed by the command, such as `w` to write the file, and press Enter.

Although `vi` supports many complex editing operations and numerous commands, you can accomplish work by using a few basic commands. These basic `vi` commands are as follows:

- ▶ **Cursor movement**—`h, j, k, l` (left, down, up, and right)
- ▶ **Delete character**—`x`
- ▶ **Delete line**—`dd`
- ▶ **Mode toggle**—`Esc`, Insert (or `i`)
- ▶ **Quit**—`:q`
- ▶ **Quit without saving**—`:q!`
- ▶ **Run a shell command**—`:sh` (use `'exit'` to return)
- ▶ **Save file**—`:w`
- ▶ **Text search**—`/`

NOTE

Use the `vimtutor` command to quickly learn how to use `vi`'s keyboard commands. The tutorial takes less than 30 minutes, and it teaches new users how to start or stop the editor; navigate files; insert and delete text; and perform search, replace, and insert operations.

Working with `emacs`

Richard M. Stallman's GNU `emacs` editor, like `vi`, is available with Ubuntu and nearly every other Linux distribution. Unlike other UNIX and Linux text editors, `emacs` is much more than a simple text editor. It's an editing environment, and you can use it to compile and build programs and act as an electronic diary, appointment book, and calendar. Use it to compose and send email, read Usenet news, and even play games. The reason for this capability is that `emacs` contains a built-in language interpreter that uses the Elisp (`emacs` LISP) programming language. `emacs` is not installed in Ubuntu by default. To use `emacs`, the package you need to install is called `emacs`. See Chapter 9, "Managing Software."

You can start an `emacs` editing session like this:

```
matthew@seymour:~$ emacs file.txt
```

TIP

If you start `emacs` when using X11, the editor launches in its own floating window. To force `emacs` to display inside a terminal window instead of its own window (which can be useful if the window is a login at a remote computer), use the `-nw` command-line option like this: `emacs -nw file.txt`.

The `emacs` editor uses an extensive set of keystroke and named commands, but you can work with it by using a basic command subset. Many of these basic commands require you to hold down the `Ctrl` key, or to first press a *meta* key (generally mapped to the `Alt` key). The basic `emacs` commands are listed as follows:

- ▶ **Cursor left**—`Ctrl+B`
- ▶ **Cursor down**—`Ctrl+N`
- ▶ **Cursor right**—`Ctrl+F`
- ▶ **Cursor up**—`Ctrl+P`
- ▶ **Delete character**—`Ctrl+D`
- ▶ **Delete line**—`Ctrl+K`
- ▶ **Go to start of line**—`Ctrl+A`
- ▶ **Go to end of line**—`Ctrl+E`
- ▶ **Help**—`Ctrl+H`
- ▶ **Quit**—`Ctrl+X`, `Ctrl+C`
- ▶ **Save as**—`Ctrl+X`, `Ctrl+W`
- ▶ **Save file**—`Ctrl+X`, `Ctrl+S`
- ▶ **Search backward**—`Ctrl+R`
- ▶ **Search forward**—`Ctrl+S`
- ▶ **Start tutorial**—`Ctrl+H`, `T`
- ▶ **Undo**—`Ctrl+X`, `U`

One of the best reasons to learn how to use `emacs` is that you can use nearly all the same keystrokes to edit commands on the `bash` shell command line, although it is possible to change the default to use `vi` key bindings. Another reason is that like `vi`, `emacs` is universally available for installation on nearly every UNIX and Linux system, including Apple's `macOS`.

Working with `sed` and `awk`

`sed`, which is short for *stream editor*, is a command that is used to perform transformations on text. It works from the command line and processes text via standard in and standard out. It does not modify the original input and does not save the output unless you redirect that output to a file. It is most useful this way or when piped between other commands.

`awk` is a small programming language for processing strings. It takes in text, transforms it in whatever way you tell it to, and then outputs the transformed text. It doesn't do as much as other languages, but what it does do it does with elegance and speed.

Both `sed` and `awk` run from the command line. There is a lot to them—more than can be covered in a short section of one chapter of a long book—but you can learn enough about each in just a few minutes to find them useful. If you want to continue learning about them, great resources are available online and in the aptly named book *sed & awk* by Dale Dougherty and Arnold Robbins.

`sed` and `awk` aren't used much anymore, at least not by people who have entered the profession in the twenty-first century, but they are beloved by those who take the time to learn them. For this edition of the book, we are including only a brief mention and a couple quick examples—certainly not enough to really learn to use either in a large capacity. If there is significant interest, we may break this out and expand it into a separate chapter in a future edition.

You use `sed` with `sed` commands, like this:

```
matthew@seymour:~$ sed sedcommand inputfile
matthew@seymour:~$ sed -e sedcommand inputfile
matthew@seymour:~$ sed -e sedcommand -e anothersedcommand inputfile
```

The second example does the same thing as the first example, except that it specifically denotes the command to run. This is optional when there is only one command, but it is useful when you want to run other commands, as in the third example.

Let's say you want to change every instance of *camel* in the text file `transportation.txt` to *dune buggy*. Here is how you do that:

```
matthew@seymour:~$ sed -e 's/camel/dune buggy/g' transportation.txt
```

The `s` command stands for *substitution*. It is surrounded by `'` marks to prevent confusion with spaces or other characters in the strings it contains. The `s` is followed by `/` (slash), which is a delimiter separating the parts of the command from one another. Next is the pattern to match, followed by what it will be changed to. Finally, the letter `g` means to replace it globally, or everywhere in the text file that *camel* occurs.

You can process text based on line numbers in the file. If you wanted to delete lines 4 through 17 in the file `longtext.txt`, you would do this:

```
matthew@seymour:~$ sed -e '4,17d' longtext.txt
```

The characters used for `sed` commands are generally obvious, like the `d` in this example standing for *delete*. You can use a regular expression in place of the line numbers. You can also use other commands besides substitute and delete. You can use `sed` in scripts and chain commands together.

The most common use of `awk` is to manipulate files that consist of fields separated by delimiters, such as a comma-separated values (CSV) file output from a spreadsheet program or a configuration file that assigns default values to program variables.

You define the delimiter that `awk` will look for, and it then assigns an internal `awk` variable to each item on a line. That is, if you have a set of parameters and values in a file where

each parameter is listed, followed by an equals sign as the delimiter, then a value, you define this for `awk` in your command, and the parameter will be assigned, for example, as `$1` and the value as `$2`.

Most files contain more than lists of parameters and values, though. What if you had a comma-delimited file containing names of things on your desk, a category for each, a color for each, and a date corresponding to the last time you picked up each item? That is four columns: name, category, color, and date. If you only really cared about the names and dates, you could use `awk` to process the file quickly and list just these, like this:

```
matthew@seymour:~$ awk -F',' '{print $1, "was last picked up on", $4}' desk-
stuff.txt
```

The output would be displayed on the screen (but could be redirected to a file) and would contain a list of only the information you wanted. In the preceding command, `-F` defines the delimiter, which is placed in `'` marks, and the pair of `{ }` within a set of `'` marks defines what to output—first variable `1`, then the text `was last picked up on`, followed by variable `4`. At the end, the text file to process is named.

You can define multiple delimiters by using `[]`, like this: `-F' [; , -] '`. You can adjust how to format the text that is output. You can output placeholders when there are blank variables. You can place several `awk` statements in a file and then run it as if it were a shell script.

To close this introduction, say that you have a large text file containing delimited lists of data, and that file contains far more information than you need. After you extract the data you need from the file, you want to replace a subset of that data with a different value. Give yourself a minute to think at a high level about how you might be able to process the file through `awk`, then pipe it into `sed`, and then redirect the output to a file. Think about how long that would take you to perform by hand or even with most programming languages, like Python or Perl. Consider how long those programs would be and, in the case of Perl, how difficult it might be to read it later. Now you know why people who know them love `sed` and `awk`.

Working with Compressed Files

Another file management operation is compression and decompression of files, or the creation, listing, and expansion of file and directory archives. Linux distributions usually include several compression utilities you can use to create, compress, expand, or list the contents of compressed files and archives. These commands include the following:

- ▶ `bunzip2`—Expands a compressed file
- ▶ `bzip2`—Compresses or expands files and directories
- ▶ `gunzip`—Expands a compressed file
- ▶ `gzip`—Compresses or expands files and directories

- ▶ **tar**—Creates, expands, or lists the contents of compressed or uncompressed file or directory archives known as *tape archives* or *tarballs*
- ▶ **xz**—Creates or expands files or *directories*

Most of these commands are easy to use. However, the `tar` command, which is the most commonly used of the bunch, has a somewhat complex set of command-line options and syntax. This flexibility and power are part of its popularity: You can quickly learn to use `tar` by remembering a few of the simple command-line options. For example, to create a compressed archive of a directory, use `tar`'s `czf` options, like this:

```
matthew@seymour:~$ tar czf compressedfilename.tgz directoryname
```

The result is a compressed archive (a file ending in `.tgz`) of the specified directory (and all files and directories under it). Add the letter `v` to the preceding options to view the list of files added during compression and archiving while the archive is being created. To list the contents of the compressed archive, substitute the `c` option with the letter `t`, like this:

```
matthew@seymour:~$ tar tzf archive
```

However, if many files are in the archive, a better invocation (to easily read or scroll through the output) is this:

```
matthew@seymour:~$ tar tzf archive | less
```

To expand the contents of a compressed archive, use `tar`'s `zxf` options, as follows:

```
matthew@seymour:~$ tar zxf archive
```

The `tar` utility decompresses the specified archive and extracts the contents in the current directory.

Using Multiple Terminals with `byobu`

Many Linux veterans have enjoyed and use the GNU `screen` command, which was designed to enable you to use one terminal to control several terminal sessions easily (see www.gnu.org/software/screen/ for more info). Although `screen` has been and is a welcome and useful tool, you should consider `byobu`, which is an enhanced version of `screen`. *Byobu* is a Japanese term for decorative, multipanel, vertically folding screens that are often used as room dividers.

Picture this scene: You connect to a server via *Secure Shell (SSH)* and are working at the remote shell. You need to open another shell window so you can have the two running side by side; perhaps you want the output from `top` in one window while you're typing in another. What do you do? Most people would open another SSH connection, but that is both wasteful and unnecessary. Like `screen`, `byobu` is a terminal multiplexer, which is a fancy term for a program that enables you to run multiple terminals inside one terminal.

The best way to learn `byobu` is to try it yourself. So, open a console, type `byobu`, and then press Enter. Your display blinks momentarily and is then replaced with a new console with new information in a panel at the bottom. Now do something with that terminal: Run `top` and leave it running for the time being. Press F2. Your prompt clears again, leaving you able to type. Run the `uptime` command.

Pop quiz: What happened to the old terminal running `top`? It is still running, of course. You can press F3 to return to it. Press F4 to go back to your `uptime` terminal. While you are viewing other terminals, the commands in the other terminals carry on running as normal so you can multitask. Here are some of the basic commands in `byobu`:

- ▶ **F2**—Creates a new window
- ▶ **F3**—Goes to the previous window
- ▶ **F4**—Goes to the next window
- ▶ **F9**—Opens the `byobu` menu for help and configuration

To close a terminal within `byobu`, simply log out of it normally, using `exit` or Ctrl+D. When you exit the last terminal session that is open in `byobu`, the program closes as well and drops you to the regular terminal session you used to start `byobu`.

However, there are two alternatives to quitting a `byobu` session: locking and disconnecting. The first, activated with F12, locks access to your screen data until you enter your system password.

The second is the most powerful feature of `screen` and also works beautifully in `byobu`: You can exit it and do other things for a while and then reconnect later; both `screen` and `byobu` allow you to pick up where you left off. For example, you could be typing at your desk, detach from a session and go home, reconnect, and carry on as if nothing had changed. What's more, all the programs you ran from `screen` or `byobu` carry on running even while `screen` or `byobu` is disconnected. They even automatically disconnect for you if someone closes your terminal window while it is in a locked state (with Ctrl+A+X).

To disconnect, press F6. You are returned to the prompt from which you launched `screen` or `byobu` and can carry on working, close the terminal you had opened, or even log out completely. When you want to reconnect, run the command `screen -r` or `byobu -r`. You can, in the meantime, just run `screen` or `byobu` and start a new session without resuming the previous one, but that is not wise if you value your sanity. You can disconnect and reconnect the same session as many times you want, which potentially means you need never lose your session again.

Although this has been a mere taste of what `byobu` and `screen` can do, hopefully you can see how useful they can be. Check the man pages for each to learn more. You can also find `byobu` documentation at <https://byobu.co> and <https://help.ubuntu.com/community/Byobu>.

Doing a Polite System Reset Using REISUB

Sometimes computer systems freeze. We're not talking about the times when one program starts acting weird and the program freezes and everything else works fine. In those cases, you can use `kill` to terminate the program and move on, as described in the section "Printing Resource Usage with `top`," earlier in this chapter or you can use the `kill` command as described in Chapter 16. The freezing we're talking about is when nothing will work. Nothing responds to any keyboard or other input, not even a Ctrl+Alt+Del key combination. What then? The absolute worst-case scenario is to perform a power cycle, which is a fancy way of saying "Turn it off and back on again." The problem is that power cycling can cause you to lose data because it can corrupt the file system. This doesn't always happen, but it is a large enough risk that you want to avoid performing a power cycle unless absolutely necessary. Instead, you can try using REISUB.

NOTE

Before you can use the REISUB feature, it must be enabled. The feature is enabled when the value of `/proc/sys/kernel/sysrq` is set to 1. You must have this enabled before you encounter a problem in order for REISUB to work.

To check the current value of this variable, run the following:

```
matthew@seymour:~$ cat /proc/sys/kernel/sysrq
```

To change the value, first edit the `/etc/sysctl.conf` file by uncommenting the line in the file by removing the # in front of it and saving to set `kernel.sysrq=1`. Then run the following:

```
matthew@seymour:~$ sudo sysctl -p /etc/sysctl.conf
```

The Linux kernel has a set of key combination commands that are built in at the kernel level. These are referred to using the name of one of the keys, the SysRq key, often labeled PrtScr. The *Magic SysRq Key* combinations send commands directly to the kernel, bypassing any programs running on top of the kernel, including your window manager and probably anything that is frozen. To use these commands, press SysRq+Alt+ one other key. Here we focus on the six keys that are most useful to most people; you can find full lists of available commands at www.kernel.org/doc/html/latest/admin-guide/sysrq.html.

REISUB is an acronym used as a mnemonic device to help users remember the Magic SysRq Key sequence that is best to use when trying to restart a frozen system without risking damage to the file system. You hold down SysRq+Alt and press the R, E, I, S, U, B keys one at a time, in order. This performs the following series of actions, listed in order here, with the letter corresponding to the command capitalized:

- ▶ **unRaw**—Takes control of the keyboard back from the X server
- ▶ **tErminate**—Sends a SIGTERM command to all processes, which allows time for the processes to terminate gracefully

- ▶ **kIII**—Sends a SIGKILL to all processes, forcing any that are still running to terminate immediately
- ▶ **Sync**—Flushes data from memory to disk
- ▶ **Unmount**—Unmounts and remounts all file systems as read-only
- ▶ **reBoot**—Turns off and back on again, restarting the computer

If you have to use REISUB, allow several seconds for each step. Be patient. Doing it this way can save you from the heartache of lost data.

Fixing an Ubuntu System That Will Not Boot

Although it's uncommon, it happens: Sometimes a system will not boot. There are many reasons a system won't boot. The goal here is to help you discover one that may help you recover your system. The ideas in this section are for computers that have had a working Ubuntu installation; however, they may also be useful if you attempted an install that did not work. The options described here are not going to help with troubleshooting computers running Windows or other operating systems.

Checking BIOS

If your computer is unable to boot at all—not even from a known-good bootable USB drive or live DVD—there are two options. It is possible that you accidentally reset the boot devices and/or order in your system BIOS. If making sure those settings are correct does not help, you may have a hardware problem.

Checking GRUB

If you are able to turn on the computer on and get past the initial BIOS startup, then you should consider whether you can access GRUB. As discussed in greater detail in Chapter 15, “The Boot Process,” the GRUB boot loader takes over after the BIOS has completed its initial work. Hold down the Shift key after the BIOS part is done to bring up the GRUB menu. If GRUB does not appear, then perhaps it has been overwritten, in which case the next section will help. If GRUB is working fine, skip to the “Using Recovery Mode” section.

Reinstalling GRUB

To restore GRUB, follow these steps:

NOTE

If you have a dual-boot system, you must be extremely careful with the steps in this section because the details in step 2 may differ depending on the other operating system residing on your system. Troubleshooting dual-boot systems is beyond the scope of this book.

1. Boot Ubuntu from a live DVD or bootable USB drive that has the same Ubuntu release as your system, such as 16.04.
2. Determine the boot drive on your system:
 - a. Open a terminal and use `sudo fdisk -l` to list the drives attached to the system.
 - b. Look for an entry in the output with an `*` in the `Boot` column. This is your boot device. It will look something like `/dev/sda1` or `/dev/nvmen0p1`.
3. Mount the Ubuntu partition at `/mnt` by using this command, replacing `/dev/sda1` with the information you just found:


```
matthew@seymour:~$ sudo mount /dev/sda1 /mnt
```
4. Reinstall GRUB with this command, again replacing `/dev/sda1` with what you found earlier:


```
matthew@seymour:~$ sudo grub-install --boot-directory=/mnt/boot /dev/sda1
```
5. Restart the computer, and Ubuntu should boot properly.

Using Recovery Mode

If GRUB is already working but you are unable to access Ubuntu, you may be able to use recovery mode. Press `Shift` after the BIOS is done to access the GRUB menu. Select `Advanced Options for Ubuntu`. From the new menu, select an entry with the words *recovery mode*. This boots into a recovery menu with options to automatically fix several possible problems, or at least it lets you boot into a minimal recovery-mode version of Ubuntu with only the most necessary processes loaded. From here, you may be able to fix disks, check file systems, drop to a root prompt to fix file permissions, and so on. If you don't understand the entries in this menu, they aren't likely to help you much, and you should consider the next option.

Reinstalling Ubuntu

If you are able to boot using a live DVD or bootable USB drive using the same Ubuntu release or one just newer than the one on the hard drive, and if there are no hardware problems with your system, you can usually recover all your files by reinstalling Ubuntu. Boot from the install medium and select `Install Ubuntu`. Make sure you are paying attention. The installer will detect an existing Ubuntu installation and give you the option to reinstall Ubuntu. When you do this, it should not overwrite existing files in your `/home` directory. Note that we said *should* not—not *will* not—and you should consider this an option of last resort.

Tips and Tricks

This last section is a motley collection of useful command-line tidbits that don't really fit well in the other categories but that are worth sharing. Enjoy.

Running the Previous Command

You can rerun the previous command with the up arrow and `Enter`. You can also rerun it with `!!` (referred to as “bang bang”). This is especially useful for times when you typed a command correctly but forgot to preface it with `sudo`, as shown here:

```
matthew@seymour:~$ apt-get update
E: Could not open lock file /var/lib/apt/lists/lock - open (13: Permission denied)
```

```
E: Unable to lock directory /var/lib/apt/lists/
E: Could not open lock file /var/lib/dpkg/lock - open (13: Permission denied)
E: Unable to lock the administration directory (/var/lib/dpkg/), are you root?
matthew@seymour:~$ sudo !!
```

This runs `sudo apt-get update`.

Running Any Previous Command

You found a neat trick and ran it, but you can't remember it. That is frustrating. You know you can use the up and down arrows to search through your command history and try to find it, but you last ran it earlier in the week, and you aren't sure how long that will take. No worries. You can search your command history.

Type `Ctrl+R` at the command line to start what is called a “reverse-i search” and begin typing. Whatever you type will be matched to previously run commands, so if you know it was a cool combination of commands piped together that had `sort` in the middle, start typing “`sort`” and watch as the displayed commands from your history appear. When you find it, press `Enter` and run it again.

Running a Previous Command That Started with Specific Letters

Say that you are listing the contents of a directory that is several layers deep, and you last ran it about nine commands ago, but you don't feel like scrolling. No sweat. Use `!` (exclamation point) and the letters that make up that command or its beginning, as shown here:

```
matthew@seymour:~$ !ls
```

This runs the most recently run command that started with `ls`.

Running the Same Thing You Just Ran with a Different First Word

Say that you just used `ls` to list a file, and you have confirmed that it is present. Now you want to use `nano` to edit the file. Use `!*` (exclamation point asterisk):

```
matthew@seymour:~$ ls stuff/article.txt
article.txt
matthew@seymour:~$ nano !*
```

Viewing Your History and More

By default, the previous 1,000 commands you have run are saved in your `/home` directory in a file called `.bash_history`. You can edit this file. You can delete this file. You can change the number of commands saved in your history by editing this line in the `.bashrc` file in your `/home` directory to whatever number you want:

```
HISTSIZE=1000
```

Doing Two or More Things

There are a few ways you can do two or more things on one line.

Separating commands with a `;` (semicolon) causes the second command to execute after the first command is complete, regardless of the result of the first command. Here is an example:

```
matthew@seymour:~$ command1 ; command2
```

If you want the second command to be run only if the first command exited with no errors, use `&&` (two ampersands):

```
matthew@seymour:~$ command1 && command2
```

If you want the second command to be run only if the first command fails, use `||` (two pipes):

```
matthew@seymour:~$ command1 || command2
```

Using Shortcuts

We all make typing errors while entering commands. When you notice an error in a long line of text, just before you press Enter, it is frustrating to use the Backspace key to erase the entire line one character at a time. There are faster ways.

- ▶ **Ctrl+U**—Erases the entire line
- ▶ **Ctrl+W**—Erases word by word
- ▶ **Left and right arrow keys**—Move along the line to where the error is
- ▶ **Ctrl+A**—Moves the cursor to the beginning of the line
- ▶ **Ctrl+E**—Moves the cursor to the end of the line
- ▶ **Ctrl+K**—Erases everything to the right of the cursor's position
- ▶ **Ctrl+Y**—Restores something you just deleted but shouldn't have

Confining a Script to a Directory

Sometimes you want to isolate a process from the rest of the system, such as when you want to test a script you have written, but you also want to make sure the script is only able to affect what you want it to and not anything else. To do this, you can set up what is called a `chroot jail`. Really, all you are doing is creating a new directory, copying the files you need for the process to run into that directory, and then using the `chroot` command to change the root directory to the base of this new file system tree. Explained differently, you are making the system act temporarily as if the directory you just named is root, when in reality nothing in the file system has changed.

For example, let's say you have a simple file system like this:

```

/
├etc
├home
│ └testing
│   └fakeetc
│     └www
└var
  └www

```

If you enter this:

```
matthew@seymour:~$ chroot testing
```

And follow it with this:

```
matthew@seymour:~$ ls /
```

You receive this output:

```

/
└fakeetc
  └www

```

TIP

It is possible for processes that run as root to “break out” of a `chroot` environment, so maybe `chroot jail` is not the most accurate term, but it is commonly used. It is better to think of this as a means to do some isolated testing but not truly secure testing.

Using Coreutils

You have already learned about some of the contents of a package of useful command-line tools called GNU Coreutils. It includes some of the most commonly used commands, like `ls`, `mv`, `cp`, `rm`, and `cat`. It also contains a ton of lesser-known but incredibly useful tools. This package is installed by default. Few people ever make use of its richness. You will want to explore it more deeply. Coreutils contains so much, it is worthy of a chapter and maybe a book of its own. What we can do here is point you to the GNU website entry for Coreutils, at www.gnu.org/software/coreutils/, and also the info page at `info coreutils` (there is no man page for Coreutils).

Coreutils includes so many useful commands, many of which are covered across this book's command-line chapters. For a version of the Coreutils manual that is easier to read than the `info` page, see <https://www.gnu.org/software/coreutils/manual/coreutils.html>.

Reading the Contents of the Kernel Ring Buffer with `dmesg`

Although it sounds fancy and ominous, the *kernel ring buffer* is actually quite simple—at least conceptually. It records a limited set of messages related to the operation of the Linux kernel. When it reaches a certain number of messages, it deletes the oldest message every time a new message is written to the list. Looking at the contents of the kernel ring buffer with `dmesg` can be helpful in determining what is happening with your system. If you enter the following, you will receive an incredibly long and scary-looking list of data:

```
matthew@seymour:~$ dmesg
```

This is too much information to be really helpful. The best use of `dmesg`, then, is to combine it with a filter and look for specific entries in the output. You might search for a part of the system, like “memory,” or a specific vendor, like “nvidia,” to read messages containing that text, like so:

```
matthew@seymour:~$ dmesg | grep nvidia
```

References

- ▶ **www.vim.org**—Home page for the `vim` (`vi` clone) editor included with Linux distributions, which provides updates, bug fixes, and news about this editor
- ▶ **www.gnu.org/software/emacs/emacs.html**—Home page for the FSF’s GNU `emacs` editing environment, where you can find additional documentation and links to the source code for the latest version
- ▶ **www.nano-editor.org**—Home page for the GNU `nano` editor environment
- ▶ ***sed & awk* by Dale Dougherty and Arnold Robbins**—The standard book for learning `sed` and `awk`
- ▶ ***The UNIX CD Bookshelf***—Seven highly respected books in one, which retails for more than \$120 but is incomparable in its depth and breadth of coverage

CHAPTER 13

Managing Users

System administrators would have a boring, but much easier, life without users. In reality, having a system with absolutely no users is impossible, so learning how to effectively manage and administer your users as they work with your system is important. Whether you are creating a single user account or modifying a group that holds hundreds of user accounts, the fundamentals of user administration are the same.

User management and administration includes allocating and managing `/home` directories, putting in place good password policies, and applying effective security policies that include things such as disk quotas and file and directory access permissions. This chapter covers all these areas as well as some of the different types of users that you are likely to find on a typical Linux system.

User Accounts

You normally find three types of users on Linux systems: the super user, the day-to-day user, and the system user. Each type is essential to the smooth running of your system. Learning the differences between the three is essential if you are to work efficiently and safely within your Linux environment.

All users who access your system must have accounts on the system. Ubuntu uses the `/etc/passwd` file to store information on the user accounts that are present on the system. All users, regardless of their type, have a one-line entry in this file that contains their username (typically used for logging in to the system), an encrypted field for the password (which contains an `X` to indicate that a password is present), a user ID (commonly referred to as the UID), and a group ID (commonly referred to as the GID).

IN THIS CHAPTER

- ▶ User Accounts
- ▶ Managing Groups
- ▶ Managing Users
- ▶ Managing Passwords
- ▶ Granting System Administrator Privileges to Regular Users
- ▶ Disk Quotas
- ▶ Related Ubuntu Commands
- ▶ References

The last two fields show the location of the `/home` directory (usually `/home/username`) and the default shell for the user (`/bin/bash` is the default for new users). There is also a field called GECOS that uses a comma-delimited list to record information about the account or the user; most often when this field is used, it records the user's full name and contact information.

NOTE

Although the Password field contains an `X`, this doesn't mean that what you read here is the actual password. All passwords are stored in `/etc/shadow` in an encrypted format for safekeeping. Ubuntu automatically refers to this file whenever a password is required. You can read more about this later in the chapter, in the "Shadow Passwords" section.

In keeping with long-standing tradition in UNIX-style operating systems, Ubuntu makes use of the well-established UNIX file ownership and permission system. To start with, everything in these systems is treated as a file, and all files (which can include directories and devices) can be assigned one or more read, write, and execute permissions. These three "flags" can also be assigned as desired to each of three categories: the owner of the file, a member of a group, or anyone else on the system. The security for a file is drawn from these permissions and from file ownership. As the system administrator (also commonly referred to as the *super user*), it is your responsibility to manage these settings effectively and ensure that the users have proper UIDs and GIDs. Perhaps most importantly, the system administrator can use these file permissions to lock away sensitive files from users who should not have access to them.

The Super User/Root User

No matter how many system administrators there are for a system, there can be only one super user account. The super user account, more commonly referred to as the *root user*, has total and complete control over all aspects of the system. That account can access any part of the file system; read, change, or delete any file; grant and revoke access to files and directories; and carry out any operation on the system, including destroying it. The root user is unique in that it has a UID of 0 and GID of 0.

Because the root user has supreme power over a system, it's important that you not work as root all the time because you might inadvertently cause serious damage to your system, perhaps even making it totally unusable. Instead, work as root only when you need to make specific changes to your system that require root privileges. As soon as you've finished that work, you can switch back to your normal user account to carry on working.

In Ubuntu, you execute a command with root, or super user, privileges by using the `sudo` command, like this:

```
matthew@seymour:~$ sudo apt-get update
```

You are then prompted for your password, which does not show on the screen as you enter it. After typing in your password, press Enter. Ubuntu then carries out the command

(in this case updating information about available software) as if you were running it as root.

THE ROOT USER

If you've used other Linux distros, you might be a little puzzled by the use of the `sudo` command because not all distros use it. In short, Ubuntu allows the first user on the system access to full root privileges using the `sudo` command. It also disables the root account so that no one can actually log in with the username `root`.

In other Linux distros, you change to the root user by issuing the command `su -` and then entering the root password when prompted. This lands you at the root prompt, which is shown as a pound sign (`#`). From here, you can execute any command you want. To get to a root prompt in Ubuntu, you need to execute the command `sudo -i`, and after you enter your password, you get the prompt so familiar to other Linux distros. When you've finished working as root, type `exit` and press Enter to get back to a normal user prompt (`$`).

A regular user is someone who logs on to the system to use it for nonadministrative tasks such as word processing or email. These users do not need to make system-wide changes or manage other users. However, they might want to be able to change settings specific to their accounts (for instance, a desktop background). Depending on how much control the system administrator (the root or super user) likes to wield, regular users might not even be able to do that.

The super user grants privileges to regular users using file and directory permissions (as covered in Chapter 10, "Command-Line Beginner's Class"). For example, if the super user does not want you to change your settings in `~/.profile` (the `~` is a shell shortcut representing your `/home` directory), then as root she can alter the permissions so that you may read from, but not write to, that file.

CAUTION

Because of the potential for making a catastrophic error as the super user, use your system as a regular user most of the time and use your super user powers only temporarily to do specific administration tasks. This is easier to remember in Ubuntu than in some other distros because the root account is initially disabled, and you must manually choose to use `sudo`. If you work on a multiuser system, consider this advice an absolute rule; if root were to delete the wrong file or kill the wrong process, the results could be disastrous for the system (and likely the business that owns and operates it). On your home system, you can do as you please. Running as root makes many things easier—but much less safe—and we still do not recommend using it all the time. In any setting, the risks of running as root are significant, and we cannot stress enough how important it is to be careful when working as root.

The third type of user is the system user. The system user is not a person but rather an administrative account that the system uses during day-to-day running of various services. For example, the system user named `www-data` owns the Apache web server and all the associated files. Only that user and root can have access to these files; no one else can

access or make changes to these files. System users do not have a home directory or password, nor do they permit access to the system through a login prompt.

You can find a list of all the users on a system in the `/etc/passwd` file.

User IDs and Group IDs

A computer is, by its very nature, a number-oriented machine. It identifies users and groups by numbers known as the *user ID (UID)* and *group ID (GID)*. The alphabetic names display on your screen just for ease of use.

As previously mentioned, the root user is UID 0. Numbers from 1 through 499 and number 65,534 are the system, sometimes called logical users, or pseudo-users. Regular users have UIDs beginning with 1,000; Ubuntu assigns them sequentially, beginning with this number.

With only a few exceptions, the GID is the same as the UID.

Ubuntu creates a private GID for every UID of 1,000 and greater. The system administrator can add other users to a GID or create a totally new group and add users to it. Unlike in Windows NT and some UNIX variants, a group in Ubuntu (or any other Linux distribution) cannot be a member of another group.

File Permissions

There are three types of permissions: read, write, and execute (*r, w, x*). For any file or directory, permissions are assigned to three categories: user, group, and other. This section focuses on group permissions. First, though, we want to highlight three commands used to change the group, user, or access permissions of a file or directory:

- ▶ **chgrp**—Changes the group ownership of a file or directory
- ▶ **chown**—Changes the owner of a file or directory
- ▶ **chmod**—Changes the access permissions of a file or directory

You can use these commands to reproduce organizational structures and permissions in the real world in your Ubuntu system (see the next section, “Managing Groups”). For example, a human resources department can share health-benefit memos to all company employees by making the files readable (but not writable) by anyone in an accessible directory. Programmers in the company’s research and development section, although able to access each other’s source code files, would not have read or write access to HR pay scale or personnel files—and certainly would not want HR or marketing poking around R&D.

These commands are used to easily manage group and file ownerships and permissions from the command line. It is essential that you know these commands because there are times when you are likely to have only a command-line interface to work with. (There was a time when a well-meaning but fat-fingered system administrator set incorrect permissions on X11, rendering the system incapable of working with a graphical interface. No, we won’t tell that story, but if you press them, most systems administrators have similar tales of woe.)

USER STEREOTYPES

As is the case in many other professions, exaggerated stereotypes have emerged for users and system administrators. Many stereotypes contain elements of truth mixed with generous amounts of hyperbole and humor and serve to assist us in understanding the characteristics of and differences in the stereotyped subjects. The stereotypes of the *luser* and the *BOFH* (not-so-nice terms for users and administrators, respectively) serve as cautionary tales describing what behavior is acceptable and unacceptable in the computing community.

Understanding these stereotypes enables you to better define the appropriate and inappropriate roles of system administrators, users, and others. You can find descriptions of these terms on Wikipedia at <https://en.wikipedia.org/wiki/Luser> and <https://en.wikipedia.org/wiki/BOFH>.

Managing Groups

Groups can make managing users a lot easier. Instead of having to assign individual permissions to every user, you can use groups to grant or revoke permissions to a large number of users quickly and easily. Setting group permissions enables you to set up workspaces for collaborative working and to control what devices can be used, such as external drives or DVD writers. This approach also represents a secure method of limiting access to system resources to only those users who need them. As an example, the system administrator could put the users `matthew`, `ryan`, `sandra`, `holly`, `debra`, and `mark` in a new group named `unleashed`. Those users could each create files intended for their group work and `chgrp` those files to `unleashed`.

Now, everyone in the `unleashed` group—but no one else except `root`—can work with those files. The system administrator would probably create a directory owned by that group so that its members could have an easily accessible place to store those files. The system administrator could also add other users such as `chris` and `shannon` to the group and remove existing users when their part of the work is done. The system administrator could make the user `matthew` the group administrator so that `matthew` could decide how group membership should be changed. You could also put restrictions on the DVD writer so that only `matthew` could burn DVDs, thus protecting sensitive material from falling into the wrong hands.

Group Listing

Different UNIX operating systems implement the group concept in various ways. Ubuntu uses a scheme called *UPG* (*user private group*) in which the default is that each user is assigned to a group with his or her own name. (The user's username and group name are identical.) All the groups on a system are listed in `/etc/group` file.

Here is an example of a portion of the `/etc/group` file:

```
matthew@seymour:~$ cat /etc/group
root:x:0:
daemon:x:1:
```

```

bin:x:2:
sys:x:3:
adm:x:4:matthew
tty:x:5:
disk:x:6:
mail:x:8:
news:x:9:
fax:x:21:matthew
voice:x:22:
cdrom:x:24:matthew
floppy:x:25:matthew
tape:x:26:matthew
www-data:x:33:
crontab:x:107:
ssh:x:109:
admin:x:115:matthew
saned:x:116:
gdm:x:119:
matthew:x:1000:
ntp:x:122:

```

In this example, you see a number of groups, mostly for services (mail, news, and so on) and devices (disk and so on). As previously mentioned, the system services groups allow those services to have ownership and control of their files. For example, adding Postfix to the `mail` group enables the Postfix application to access `mail`'s files such that `mail` can decide about group access to its file. Adding a regular user to a device's group permits the regular user to use the device with permissions granted by the group owner. Adding user `matthew` to the group `cdrom`, for example, allows `matthew` to use the optical drive device. You learn how to add and remove users from groups in the next section.

FINDING YOUR GROUPS

You can find which groups your user account belongs to by using the `groups` command, like this:

```
matthew@seymour:~$ groups
```

```
matthew adm cdrom sudo audio dip plugdev lpadmin sambashare
```

Add a username after the command to list the groups for that user.

Group Management Tools

Ubuntu provides several command-line tools for managing groups, and it also provides graphical tools for doing so. Most experienced system administrators prefer the command-line tools because they are quick and easy to use, they are always available (even when there is no graphical user interface), and they can be included in scripts that system administrators may want to write to perform repetitive tasks.

Here are the most commonly used group management command-line tools:

- ▶ **groupadd**—This command creates and adds a new group.
- ▶ **groupdel**—This command removes an existing group.
- ▶ **groupmod**—This command creates a group name or GIDs but doesn't add or delete members from a group.
- ▶ **gpasswd**—This command creates a group password. Every group can have a group password and an administrator. Use the **-A** argument to assign a user as group administrator.
- ▶ **useradd -G**—The **-G** argument adds a user to a group during the initial user creation. (More arguments are used to create a user.)
- ▶ **usermod -G**—This command allows you to add a user to a group as long as the user is not logged in at the time.
- ▶ **grpck**—This command checks the `/etc/group` file for typos.

Let's say there is a DVD-RW device (`/dev/scd0`) on your computer that the system administrator wants a regular user named `ryan` to have permission to access. This is the process for granting `ryan` that access:

1. Add a new group with the `groupadd` command:

```
matthew@seymour:~$ sudo groupadd dvdwr
```

2. Change the group ownership of the device to the new group with the `chgrp` command:

```
matthew@seymour:~$ sudo chgrp dvdwr /dev/scd0
```

3. Add the approved user to the group with the `usermod` command:

```
matthew@seymour:~$ sudo usermod -G dvdwr ryan
```

4. Make user `ryan` the group administrator with the `gpasswd` command so that he can add new users to the group:

```
matthew@seymour:~$ sudo gpasswd -A ryan
```

Now `ryan` has permission to use the DVD-RW drive, as would anyone else added to the group by either the super user or `ryan` because he is now also a group administrator and can add users to the group.

Managing Users

A user must be created, assigned a UID, provided a `/home` directory, provided an initial set of files for his or her `/home` directory, and assigned to groups in order to use the system resources securely and efficiently. The system administrator in some situations might want or need to restrict not only a user's access to specific files and folders but also the amount of disk space an account may use.

User Management Tools

As with groups, Ubuntu provides several command-line tools for managing users, and it also provides graphical tools for doing so. As mentioned earlier, most experienced system administrators prefer the command-line tools because they are quick and easy to use, they are always available (even when there is no graphical user interface), and they can be included in scripts that system administrators may want to write to perform repetitive tasks. Here are the most common commands to manage users:

- ▶ **useradd**—This command adds a new user account to the system. Its options permit the system administrator to specify the user's `/home` directory and initial group or to create the user with the default `/home` directory and group assignments (based on the new account's username).
- ▶ **useradd -D**—This command sets the system defaults for creating the user's `/home` directory, account expiration date, default group, and command shell. See the specific options in the `useradd` man page. Used without any arguments, the `useradd` command displays the defaults for the system. The default files for a user are in `/etc/skel`.

NOTE

The set of files initially used to populate a new user's home directory is kept in `/etc/skel`. This is convenient for the system administrator because any special files, links, or directories that need to be universally applied can be placed in `/etc/skel` and will be duplicated automatically with appropriate permissions for each new user:

```
matthew@seymour:~$ ls -la /etc/skel

total 32

drwxr-xr-x  2 root root  4096 2010-04-25 12:14 .
drwxr-xr-x 154 root root 12288 2010-07-01 16:30 ..
-rw-r--r--  1 root root   220 2009-09-13 22:08 .bash_logout
-rw-r--r--  1 root root  3103 2010-04-18 19:15 .bashrc
-rw-r--r--  1 root root   179 2010-03-26 05:31 examples.desktop
-rw-r--r--  1 root root   675 2009-09-13 22:08 .profile
```

Each line provides the file permissions, the number of files housed under that file or directory name, the file owner, the file group, the file size, the creation date, and the filename.

As you can see, root owns every file here. The `useradd` command copies everything in `/etc/skel` to the new home directory and resets file ownership and permissions to the new user.

Certain user files might exist that the system administrator doesn't want the user to change; the permissions for those files in `/home/username` can be reset so that the user can read them but can't write to them.

- ▶ **deluser**—This command removes a user’s account (thereby eliminating that user’s home directory and all files it contains). There is an older version of this command, `userdel`, that previous versions of this book discussed. `deluser` is preferred because it provides finer control over what is deleted. Whereas `userdel` automatically removes both the user account and also all the user’s files, such as the associated `/home` directory, `deluser` deletes only the user account, unless you use a command-line option to tell it to do more. `deluser` includes options such as `--remove-home`, `--remove-all-files`, `--backup`, and more. See the `man` page for more information.
- ▶ **passwd**—This command updates the authentication tokens used by the password management system.

TIP

To lock a user out of his or her account, use the following command:

```
matthew@seymour:~$ sudo passwd -l username
```

This prepends an `!` (exclamation point, also called a bang) to the user’s encrypted password; the command to reverse the process uses the `-u` option.

- ▶ **usermod**—This command changes several user attributes. The most commonly used arguments are `-s` to change the shell and `-u` to change the UID. No changes can be made while the user is logged in or running a process.
- ▶ **chsh**—This command changes the user’s default shell. For Ubuntu, the default shell is `/bin/bash`, known as the Bash, or Bourne Again Shell.

Adding New Users

The command-line approach to adding a user is quite simple and can be accomplished on a single line. In the following example, the system administrator uses the `useradd` command to add the new user `sandra`:

```
matthew@seymour:~$ sudo useradd sandra -p c00kieZ4ME -u 1042
```

The command `adduser` (a variant found on some UNIX systems) and `useradd` are similar and do the same thing. This example uses the `-p` option to set the password the user requested and the `-u` option to specify her UID. (If you create a user with the default settings, you do not need to use these options.) As you can see, all this can be accomplished on one line.

The system administrator can also use the graphical interface that Ubuntu provides to add the same account as shown in the preceding command but with fewer setting options available:

1. From the menu at the upper right of the desktop, select the Settings icon, which looks like a gear (see Figure 13.1). In the Settings application, from the bottom left select Users (see Figure 13.2).

2. Click Unlock at the upper right and enter your password to authorize making changes to user accounts.
3. Click Add User at the upper right, where Unlock was, to open the Add Account window.
4. Fill in the form with the new user's name and desired username, select whether to set a password now or have the new user create a password the first time he or she logs in, and designate whether the new user is a standard user or an administrator, and click Add (see Figure 13.3).

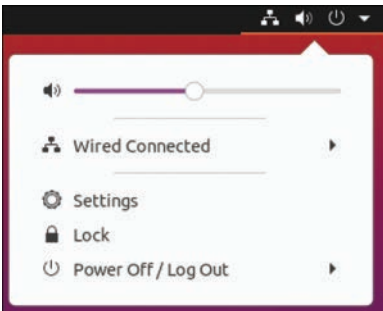


FIGURE 13.1 Open the menu at the upper right to find the Settings icon.

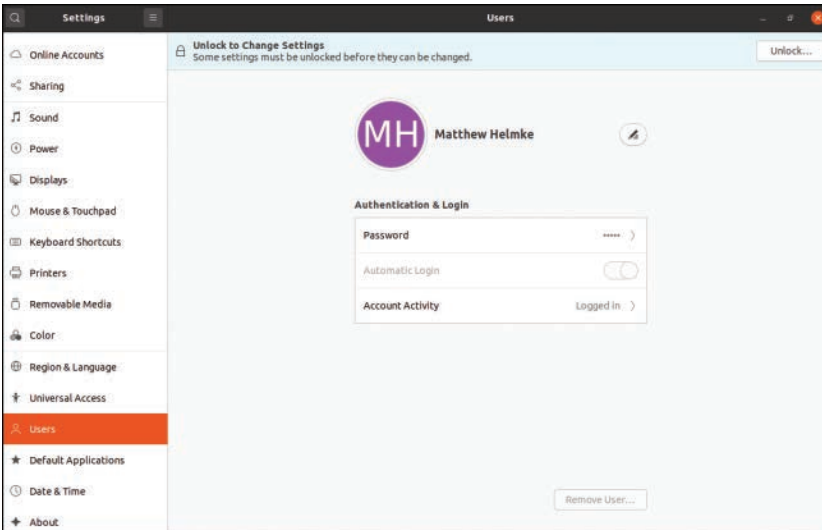


FIGURE 13.2 Select Users.

The screenshot shows a window titled "Add User" with a dark header bar containing "Cancel", "Add User", and "Add" buttons. The main content area has a light background. At the top, there are two tabs: "Standard" (selected) and "Administrator". Below the tabs are two text input fields: "Full Name" containing "Sandra Helmke" and "Username" containing "sandra". A small note below the username field states: "This will be used to name your home folder and can't be changed." Underneath is a section titled "Password" with two radio button options: "Allow user to set a password when they next login" (which is selected) and "Set a password now". Below these are three text input fields: "Password", a second "Password" field, and "Confirm". A small note between the second and third password fields says: "Mix uppercase and lowercase and try to use a number or two."

FIGURE 13.3 Adding a new user is simple. The GUI provides a set of options for user management spread over several screens.

NOTE

A Linux username can be any alphanumeric combination that does not begin with a special character reserved for shell script use (mostly `<space>` and punctuation characters; see Chapter 14, “Automating Tasks and Shell Scripting,” for disallowed characters). A username is often the user’s first name plus the first initial of her last name or the first initial of the user’s first name and his entire last name. These are common practices on larger systems with many users because it makes life simpler for the system administrator, but neither convention is a rule or a requirement.

Monitoring User Activity on the System

Monitoring user activity is part of a system administrator’s duties and an essential task in tracking how system resources are being used. The `w` command tells the system administrator who is logged in, where he is logged in, and what he is doing. No one can hide from the super user. The `w` command can be followed by a specific user’s name to show only that user.

The `ac` command provides information about the total connect time of a user, measured in hours. It accesses the `/var/log/wtmp` file for the source of its information. The `ac` command is most useful in shell scripts to generate reports on operating system usage for management review. Note that to use the `ac` command, you must install the `acct` package from the Ubuntu repositories.

TIP

Interestingly, a phenomenon known as *time warp* can occur, where an entry in the `wtmp` files jumps back into the past, and `ac` shows unusual amounts of time accounted for users. Although this can be attributed to some innocuous factors having to do with the system clock, it is worthy of investigation by the system administrator because it can also be the result of a security breach.

The `last` command searches through the `/var/log/wtmp` file and lists all the users logged in and out since that file was first created. The user `reboot` exists so that you might know who has logged in since the last reboot. A companion to `last` is the command `lastb`, which shows all failed, or bad, logins. It is useful for determining whether a legitimate user is having trouble or a hacker is attempting access.

NOTE

The accounting system on your computer keeps track of user usage statistics and is kept in the current `/var/log/wtmp` file. That file is managed by the `systemd` processes. If you want to explore the depths of the accounting system, use the GNU info system: `info accounting`.

Managing Passwords

Passwords are an integral part of Linux security, and they are the most visible part to the user. In this section, you learn how to establish a minimal password policy for your system, where the passwords are stored, and how to manage passwords for your users.

System Password Policy

An effective password policy is a fundamental part of a good system administration plan. The policy should cover the following:

- ▶ Allowed and forbidden passwords
- ▶ Frequency of mandated password changes
- ▶ Retrieval or replacement of lost or forgotten passwords
- ▶ Password handling by users

The Password File

The password file is `/etc/passwd`, and it is the database file for all users on the system. The format of each line is as follows:

```
username:password:uid:gid:gecos:homedir:shell
```

The fields are self-explanatory except for the `gecos` field. This field is for miscellaneous information about the user, such as the user's full name, office location, office and home phone numbers, and possibly a brief text note. For security and privacy reasons, this field is little used today, but the system administrator should be aware of its existence because the `gecos` field is used by traditional UNIX programs such as `finger` and `mail`. For that reason, it is commonly referred to as the *finger information field*. The data in this field is comma delimited; you can change the `gecos` field with the `chfn` (change finger) command.

Note that colons separate all fields in the `/etc/passwd` file. If no information is available for a field, that field is empty, but all the colons remain.

If an asterisk appears in the `password` field, that user is not permitted to log on. This feature exists so that a user can be easily disabled and (possibly) reinstated later without the need to create the user all over again. The traditional UNIX way of accomplishing this task is for the system administrator to manually edit this field. Ubuntu provides a more elegant method with the `passwd -l` command, mentioned earlier in this chapter.

Several services run as pseudo-users, usually with root permissions. These are the system, or logical, users mentioned previously. You would not want these accounts to be available for general login for security reasons, so they are assigned `/sbin/nologin` or `/bin/false` as their shell, which prohibits any logins from these accounts.

A list of `/etc/passwd` reveals the following (abridged for brevity):

```
matthew@seymour:~$ cat /etc/passwd
root:x:0:0:root:/root:/bin/bash
bin:x:2:2:bin:/bin:/bin/sh
sys:x:3:3:sys:/dev:/bin/sh
games:x:5:60:games:/usr/games:/bin/sh
man:x:6:12:man:/var/cache/man:/bin/sh
mail:x:8:8:mail:/var/mail:/bin/sh
news:x:9:9:news:/var/spool/news:/bin/sh
uucp:x:10:10:uucp:/var/spool/uucp:/bin/sh
www-data:x:33:33:www-data:/var/www:/bin/sh
gnats:x:41:41:Gnats Bug-Reporting System (admin):/var/lib/gnats:/bin/sh
nobody:x:65534:65534:nobody:/nonexistent:/bin/sh
messagebus:x:102:106:./var/run/dbus:/bin/false
avahi:x:105:111:Avahi mDNS daemon,,,:/var/run/avahi-daemon:/bin/false
couchdb:x:106:113:CouchDB Administrator,,,:/var/lib/couchdb:/bin/bash
haldaemon:x:107:114:Hardware abstraction layer,,,:/var/run/hald:/bin/false
kernoops:x:109:65534:Kernel Oops Tracking Daemon,,,:/bin/false
gdm:x:112:119:Gnome Display Manager:/var/lib/gdm:/bin/false
matthew:x:1000:1000:Matthew Helmke,,,:/home/matthew:/bin/bash
sshd:x:114:65534:./var/run/sshd:/usr/sbin/nologin
ntp:x:115:122:./home/ntp:/bin/false
pulse:x:111:117:PulseAudio daemon,,,:/var/run/pulse:/bin/false
```

Note that none of the `password` fields show a password but rather contain an `X`. This is because they are shadow passwords, a useful security enhancement to Linux.

Shadow Passwords

Keeping passwords in `/etc/passwd` is considered a security risk because anyone with read access could run a cracking program on the file and obtain the passwords with little trouble. To avoid this risk, *shadow passwords* are used so that only an `X` appears in the password field of `/etc/passwd`; the real passwords are kept in `/etc/shadow`, a file that can be read only by the system administrator (and *PAM*, the *Pluggable Authentication Modules* authentication manager; see the “PAM Explained” sidebar, later in this chapter, for an explanation of PAM).

Special versions of the traditional `password` and `login` programs must be used to enable shadow passwords. Shadow passwords are automatically enabled during installation of Ubuntu. Examine the following abbreviated listing of the shadow companion to `/etc/passwd`, the `/etc/shadow` file:

```
matthew@seymour:~$ sudo cat /etc/shadow
root:!:14547:0:99999:7:::
daemon*:14544:0:99999:7:::
bin*:14544:0:99999:7:::
sys*:14544:0:99999:7:::
games*:14544:0:99999:7:::
man*:14544:0:99999:7:::
mail*:14544:0:99999:7:::
www-data*:14544:0:99999:7:::
irc*:14544:0:99999:7:::
nobody*:14544:0:99999:7:::
libuuid:!:14544:0:99999:7:::
syslog*:14544:0:99999:7:::
messagebus*:14544:0:99999:7:::
kernoops*:14544:0:99999:7:::
gdm*:14544:0:99999:7:::
matthew:$6$wtML.mV4$.I5WeTp9tgGkIjJM4uLR5p6TVUqPrSvJ0N2W/t//0jVBrWQrOySEEDvXsA/sKSEL
QsfmNmFPJYxVrjZ21/Ir70:14564:0:99999:7:::
sshd*:14547:0:99999:7:::
ntp*:14548:0:99999:7:::
usbmux*:14724:0:99999:7:::
pulse*:14725:0:99999:7:::
```

The fields are separated by colons and are, in order:

- ▶ The user’s login name.
- ▶ The encrypted password for the user.
- ▶ The day on which the last password change occurred, measured in the number of days since January 1, 1970. This date is known in UNIX circles as the *epoch*.

Just so you know, the billionth second since the epoch occurred was in September 2001; that was the UNIX version of Y2K—and as with the real Y2K, nothing much happened.

- ▶ The number of days before the password can be changed (which prevents changing a password and then changing it back to the old password right away—a dangerous security practice).
- ▶ The number of days after which the password must be changed. This can be set to force the change of a newly issued password known to the system administrator.
- ▶ The number of days before the password expiration that the user is warned it will expire.
- ▶ The number of days after the password expires that the account is disabled (for security).
- ▶ Similar to the password change date, although this is the number of days since January 1, 1970, that the account has been disabled.
- ▶ A “reserved” field that is not currently allocated for any use.

Note that password expiration dates and warnings are disabled by default in Ubuntu. These features are not often used on home systems and usually are not even used for small offices. It is the system administrator’s responsibility to establish and enforce password expiration policies if they are to exist.

The permissions on the `/etc/shadow` file should be set so that it is not writable or readable by regular users: The permissions should be `600`.

PAM EXPLAINED

Pluggable Authentication Modules (PAM) is a system of libraries that handle the tasks of authentication on a computer. It uses four management groups: account management, authentication management, password management, and session management. This allows the system administrator to choose how individual applications will authenticate users. Ubuntu has preinstalled and preconfigured all the necessary PAM files for you.

The configuration files in Ubuntu are in `/etc/pam.d`. Each of these files is named for the service it controls, using the following format:

```
type control module-path module-arguments
```

The `type` field is the management group that the rule corresponds to. The `control` field tells PAM what to do if authentication fails. The final two items deal with the PAM module used and any arguments it needs. Programs that use PAM typically come packaged with appropriate entries for the `/etc/pam.d` directory. To achieve greater security, the system administrator can modify the default entries. Misconfiguration can have unpredictable results, so back up the configuration files before you modify them. The defaults provided by Ubuntu are adequate for home and small office users.

An example of a PAM configuration file with the formatted entries as described previously is shown next. Here are the contents of `/etc/pam.d/gdm`:

```
#%PAM-1.0
auth requisite pam_nologin.so
auth required pam_env.so readenv=1
auth required pam_env.so readenv=1 envfile=/etc/default/locale
auth sufficient pam_succeed_if.so user ingroup nopasswdlogin
@include common-auth
auth optional pam_gnome_keyring.so
@include common-account
session [success=ok ignore=ignore module_unknown=ignore default=bad]
[ccc]pam_selinux.so open
session required pam_limits.so
@include common-session
session [success=ok ignore=ignore module_unknown=ignore default=bad]
[ccc]pam_selinux.so close
session optional pam_gnome_keyring.so auto_start
@include common-password
```

Amusingly, even the PAM documents state that you do not really need (or want) to know a lot about PAM to use it effectively.

You will likely need only the PAM system administrator's guide. You can find it at www.linux-pam.org/Linux-PAM-html/Linux-PAM_SAG.html.

Managing Password Security for Users

Selecting appropriate user passwords is always an exercise in trade-offs. A password such as *password* (do not laugh, it has been used often in the real world and with devastating consequences) is just too easy to guess by an intruder. So are simple words or number combinations (the numbers from a street address or date of birth, for example). You would be surprised how many people use easily guessed passwords such as 123456, i_loveyou, Qwerty, and abc123.

In contrast, a password such as 2a56u' "F(\$84u&#^Hiu44Ik%\$([#EJD is sure to present great difficulty to an intruder (or an auditor). However, that password is so difficult to remember that it would be likely that the password owner would write that password down on a sticky note attached to his monitor.

The system administrator has control, with settings in the `/etc/shadow` file, over how often the password must be changed. The settings can be changed by the super user using a text editor or the `chage` command. (See the `shadow` and `chage` man pages for details.)

Changing Passwords in a Batch

On a large system, there might be times when a large number of users and their passwords need some attention. The super user can change passwords in a batch by using the

`chpasswd` command, which accepts input as a name/password pair per line in the following form:

```
matthew@seymour:~$ sudo chpasswd username:password
```

Passwords can be changed *en masse* by redirecting a list of name and password pairs to the command. An appropriate shell script can be constructed with the information gleaned from Chapters 11 and 12, “Command-Line Master Class, Part 1” and “Command-Line Master Class, Part 2,” combined with information on writing scripts from Chapter 14.

However, Ubuntu also provides the `newusers` command to add users in a batch from a text file. This command also allows a user to be added to a group, and a new directory can be added for the user, too.

Granting System Administrator Privileges to Regular Users

On occasion, regular users might need to run a command as if they were the root user. They usually do not need these powers, but a user might require such abilities on special occasions—for example, to temporarily access certain devices or run a command for testing purposes.

There are two ways to run commands with root privileges. The first way is useful if you are the owner of both the super user account (an enabled root account) and a regular user; the second way is useful if you are a regular user but are not privileged to access all super user functions. (This might happen on a large, multiuser network with senior and junior administrators as well as regular users.) Let’s look at each.

Temporarily Changing User Identity with the `su` Command

This first scenario requires the existence of a root account, which is not enabled by default on Ubuntu systems and is not generally recommended in the Ubuntu community. However, there are times when it makes sense. Discussing that is beyond the scope of this chapter, but for the sake of argument, for the scenario and details in this section, assume that you are operating in one of those special cases and that a root account has been enabled.

What if you have access to an enabled root account as a super user but are logged on as a regular user because you are performing nonadministrative tasks, and you find that you need to do something that only the super user can do? The `su` command is available for this purpose.

NOTE

A popular misconception is that the `su` command is short for *super user*; it really just means *substitute user*. An important but often overlooked distinction is that between `su` and `su -`. In the former instance, you become that user but keep your own environmental variables (such as paths). In the latter, you inherit the environment of that user. This is

most noticeable when you use `su` to become the super user, root. Without appending the `-`, you do not inherit the path variable that includes `/bin` or `/sbin`, so you must always enter the full path to those commands when you just `su` to root.

Don't forget that on a standard Ubuntu system, the first created user is classed as root, whereas the true root account is disabled. To enable the root account, you enter the command `sudo passwd` at the command line and enter your password and a new root password. After this has been completed, you can `su` to root. We suggest you read the information at <https://help.ubuntu.com/community/RootSudo> before doing so to ensure that you understand the reason the root account is not enabled by default.

Because almost all Linux file system security revolves around file permissions, it can be useful to occasionally become a different user with permission to access files belonging to other users or groups or to access special files (such as the communications port `/dev/ttyS0` when using a modem or the sound device `/dev/audio` when playing a game). You can use the `su` command to temporarily switch to another user identity and then switch back.

The `su` command spawns a new shell, changing both the UID and GID of the existing user and automatically changing the environmental variables associated with that user, known as *inheriting the environment*. For more information about environment variables, see Chapter 6, “Productivity Applications.”

The syntax for the `su` command is as follows:

```
matthew@seymour:~$ su option username arguments
```

The man page for `su` gives more details, but some highlights of the `su` command are as follows:

```
-c, --command
    pass a single COMMAND to the shell with -c

-m, --preserve-environment
    do not reset environment variables

-l  a full login simulation for the substituted user,
    the same as specifying the dash alone
```

You can invoke the `su` command in different ways that yield diverse results. By using `su` alone, you can become root, but you keep your regular user environment. This can be verified by using the `printenv` command before and after the change. Note that the working directory (you can execute `pwd` at the command line to print the current working directory) has not changed. By executing the following, you become root and inherit root's environment:

```
matthew@seymour:~$ su -
```

By executing the following, you become that user and inherit the super user's environment—a pretty handy tool. (Remember: Inheriting the environment comes from using

the dash in the command; omit that, and you keep your “old” environment.) To become another user, specify a different user’s name on the command line:

```
matthew@seymour:~$ su - other_user
```

When leaving an identity to return to your usual user identity, use the `exit` command. For example, while logged on as a regular user, use the following:

```
matthew@seymour:~$ su - root
```

The system prompts for a password:

```
Password:
```

When the password is entered correctly, the root user’s prompt appears:

```
root~#
```

To return to the regular user’s identity, just type the following:

```
root~# exit
```

This takes you to the regular user’s prompt:

```
matthew@seymour:~$
```

If you need to allow other users access to certain commands with root privileges, you must give them the password for the root account (often referred to as the root password) so that they can use `su`; that definitely is not a secure solution. The next section describes a more flexible and secure method of allowing normal users to perform selected root tasks and the preferred method for sharing and using super user privileges in Ubuntu.

NOTE

The `su` command is often seen as bad because what it is supposed to do is a bit ambiguous. On one hand, it is supposed to open a new session and change a number of execution parameters while also inheriting parameters from the session in which it was issued. It does give you a new execution shell, but that is not really the same thing as a full login. `systemd` has added a new command, `machinectl shell`, that is intended to do this “properly,” according to its creators. Because `systemd` is covered in Chapter 15, “The Boot Process,” this new command is also covered there.

Granting Root Privileges on Occasion: The `sudo` Command

Delegating some of the authority that root wields on a system is often necessary. For a large system, this makes sense because no single individual will always be available to perform super user functions. The problem is that UNIX permissions come with all-or-nothing authority. Enter `sudo`, an application that permits the assignment of one, several, or all the root-only system commands.

NOTE

As mentioned earlier, the `sudo` command is pervasive in Ubuntu because it is used by default. If you want to get to a root shell, thereby removing the need to type `sudo` for every command, just enter `sudo -i` to get the root prompt. To return to a normal user prompt, enter `exit`, and press Enter. Again, this is a bit dangerous because if you are not paying attention and forget to exit root, you could cause severe damage to the system. It is usually better to choose one method or the other and use it consistently, and the Ubuntu community consistently uses and recommends using `sudo` for each command, even if it gets tedious, because it is a good reminder to think about what you are doing.

After it is configured, using `sudo` is simple. An authorized user merely precedes a super user authority-needed command with `sudo`, like this:

```
matthew@seymour:~$ sudo command
```

When the command is entered, `sudo` checks the `/etc/sudoers` file to see whether the user is authorized to wield super user privileges; if so, `sudo` use is authorized for a specific length of time. The user is then prompted for her password (to preserve accountability and provide some measure of security), and then the command is run as if root had issued it. During the time allotted, which is 15 minutes by default in Ubuntu, `sudo` can be used again once or multiple times without a password. If an unauthorized user attempts to execute a `sudo` command, a record of the unauthorized attempt is kept in the system log, and a mail message is sent to the super user.

Three man pages are associated with `sudo`: `sudo`, `sudoers`, and `visudo`. The first covers the command itself, the second the format of the `/etc/sudoers` file, and the third the use of the special editor for `/etc/sudoers`. You should use the special editing command because it checks the file for parse errors and locks the file to prevent others from editing it at the same time. The `visudo` command uses the `vi` editor, so you might need a quick review of the `vi` editing commands found in Chapter 12 in the section “Working with `vi`.” You begin the editing by executing the `visudo` command with this:

```
matthew@seymour:~$ sudo visudo
```

The default `/etc/sudoers` file looks like this:

```
# /etc/sudoers
#
# This file MUST be edited with the 'sudo' command as root.
#
# See the man page for details on how to write a sudoers file.
#
Defaults    env_reset
Defaults    mail_badpass
Defaults    secure_path="/usr/local/sbin:/usr/local/bin:/usr/sbin:/usr/bin:/sbin:/bin:/snap/bin"
```

```
# Host alias specification

# User alias specification

# Cmnd alias specification

# User privilege specification
root    ALL=(ALL) ALL

# Allow members of group sudo to execute any command
%sudo  ALL=(All:ALL) ALL

# See sudoers(5) for more information on "#include" directives:

#include /etc/sudoers.d
```

The basic format of a `sudoers` line in the file is as follows:

```
user host_computer=command
```

The `user` can be an individual user or a group. (A `%` in front identifies a name as a group.) The `host_computer` is normally `ALL` for all hosts on the network and `localhost` for the local machine, but the host computer can be referenced as a subnet or any specific host. The command in the `sudoers` line can be `ALL`, a list of specific commands, or a restriction on specific commands (formed by prepending a `!` to the command). A number of options are available for use with the `sudoers` line, and aliases can be used to simplify the assignment of privileges. Again, the `sudoers` man page gives the details, but let's look at a few examples.

Suppose that you want to give user `john` permission across the network to be able to add users with the graphical interface. You would add the following line:

```
john ALL=/users-admin
```

Or perhaps you would grant permission only on the user's local computer:

```
john 192.168.1.87=/usr/bin/users-admin
```

If you want to give the editor group system-wide permission with no password required to delete files, you use the following:

```
%editors ALL=NOPASSWD: /bin/rm
```

If you want to give every user permission with no password required to mount the CD drive on the `localhost`, you do so as follows:

```
ALL localhost=NOPASSWD:/sbin/mount /dev/scd0 /mnt/cdrom /sbin/umount /mnt/cdrom
```

Using wildcards in the construction of the `sudoers` file is also possible. Aliases can be used, as well, to make it easier to define users and groups. The man page for `sudoers` contains

some examples, and www.komar.org/pres/sudo/toc.html provides illustrative notes and comments about `sudo` use at a large company. The `sudo` home page at www.sudo.ws is also a useful resource for additional explanations and examples.

The following command presents users with a list of the commands they are entitled to use:

```
matthew@seymour:~$ sudo -l
```

Disk Quotas

On large systems with many users, you often need to control the amount of disk space a user can use. Disk quotas are designed specifically for this purpose. Quotas, managed per partition, can be set for both individual users and groups; quotas for groups need not be as large as the aggregate quotas for the individuals in the groups.

When a file is created, both a user and a group own it. Ownership of files is always part of the metadata about the files. This makes quotas based on both users and groups easy to manage.

NOTE

Disk quota management is not really useful or needed on a home system and rarely, if ever, on a small office system. It is unlikely that you will see or implement this in either circumstance.

To manage disk quotas, you must have the `quota` and `quotatool` packages installed on your system. Quota management with Ubuntu is not enabled by default and has traditionally been enabled and configured manually by system administrators. System administrators use the family of `quota` commands, such as `quotacheck` to initialize the quota database files, `edquota` to set and edit user quotas, `setquota` to configure disk quotas, and `quotaon` or `quotaoff` to control the service. (Other utilities include `warnquota` for automatically sending mail to users over their disk space usage limit.)

Implementing Quotas

Quotas are not enabled by default, even if the quota software package is installed on your Ubuntu system. When quotas are installed and enabled, you can see which partitions have either user quotas, group quotas, or both by looking at the fourth field in the `/etc/fstab` file. For example, one line in `/etc/fstab` shows that quotas are enabled for the `/home` partition:

```
/dev/hda5    /home    ext3        defaults,usrquota,grpquota 1 1
```

The root of the partition with quotas enabled will have the file `quota.user` or the file `quota.group` in it (or both files, if both types of quotas are enabled), and the files will contain the actual quotas. The permissions of these files should be `600` so that users cannot read or write to them. (Otherwise, users would change them to allow ample space for their music files and Internet art collections.) To initialize disk quotas, the partitions must be remounted. This is easily accomplished with the following:

```
matthew@seymour:~$ sudo mount -o ro,remount partition_to_be_remounted mount_point
```

The underlying console tools (all of which have man pages) are as follows:

- ▶ **quotaon** and **quotaoff**—Toggle quotas on a partition
- ▶ **repquota**—Provides a summary status report on users and groups
- ▶ **quotacheck**—Updates the status of quotas (compares new and old tables of disk usage); run after `fsck`
- ▶ **edquota**—Enables basic quota management

Manually Configuring Quotas

Manual configuration of quotas involves changing entries in your system's file system table, `/etc/fstab`, to add the `usrquota` mount option to the desired portion of your file system. As an example, in a simple file system, you can enable quota management like this:

```
LABEL=/          /          ext3          defaults,usrquota          1 1
```

You can also enable group-level quotas by using the `grpquota` option. Then, as the root operator in the example of creating user quotas, you must create a file named `quota.user` in the designated portion of the file system, like so:

```
matthew@seymour:~$ sudo touch /quota.user
```

You should then turn on the use of quotas by using the `quotaon` command:

```
matthew@seymour:~$ sudo quotaon -av
```

You can then edit user quotas with the `edquota` command to set hard and soft limits on file system use. The default system editor (`vi` unless you change your `EDITOR` environment variable) is launched when editing a user's quota.

Users can find out what their quotas are with the following:

```
matthew@seymour:~$ quota -v
```

NOTE

Ubuntu does not support any graphical tools that enable you to configure disk quotas. A Quota mini-HOWTO is maintained at www.tldp.org/HOWTO/Quota.html.

Related Ubuntu Commands

You use these commands to manage user accounts in Ubuntu:

- ▶ **ac**—Provides user account statistics
- ▶ **change**—Sets or modifies user password expiration policies

- ▶ **chfn**—Creates or modifies user *finger* information in `/etc/passwd`
- ▶ **chgrp**—Modifies group memberships
- ▶ **chmod**—Changes file permissions
- ▶ **chown**—Changes file ownerships
- ▶ **chpasswd**—Modifies user passwords in batches
- ▶ **chsh**—Modifies a user’s shell
- ▶ **groups**—Displays existing group memberships
- ▶ **logname**—Displays a user’s login name
- ▶ **newusers**—Batches user management command
- ▶ **passwd**—Creates or modifies user passwords
- ▶ **su**—Executes a shell or command as another user
- ▶ **sudo**—Manages selected user execution permissions
- ▶ **useradd**—Creates, modifies, or manages users
- ▶ **usermod**—Edits a user’s login profile

References

- ▶ **<https://tldp.org/HOWTO/User-Authentication-HOWTO/>**—A HOWTO that describes how user and group information is stored and used for authentication
- ▶ **<https://tldp.org/HOWTO/Shadow-Password-HOWTO.html>**—A HOWTO that delves into the murky depths of shadow passwords and even discusses why you might not want to use them.
- ▶ **<https://tldp.org/HOWTO/Security-HOWTO/>**—A must-read HOWTO that provides a good overview of security issues, including creating accounts, file permissions, and password security.
- ▶ **<https://tldp.org/LDP/sag/html/index.html>**—A general guide, the Linux System Administrator’s Security Guide has interesting sections on limiting and monitoring users.
- ▶ **www.kernel.org/pub/linux/libs/pam**—The Pluggable Authentication Modules suite contains complex and highly useful applications that provide additional security and logging for passwords. PAM is installed by default in Ubuntu. Understanding the intricacies of PAM isn’t necessary to use it effectively.

CHAPTER 14

Automating Tasks and Shell Scripting

This chapter covers ways to automate tasks on your system by using task schedulers. This chapter also provides an introduction to the basics of creating *shell scripts*, or executable text files written to conform to shell syntax. Shell scripting and automation are useful on your local machine, but they are vital in DevOps and Site Reliability Engineering where the goal is to automate anything that can be automated to save human brain power for tasks that require imaginative thinking.

Shell scripts run like any other command under Linux and can contain complex logic or a simple series of Linux command-line instructions. You can also run other shell scripts from within a shell program. The features and functions for several Linux shells are discussed in this chapter after a short introduction to working from the shell command line. You find out how to write and execute a simple shell program using `bash`, one of the most popular Linux shells and the default shell in Ubuntu and most other distributions.

What Is a Shell?

We have used the shell since Chapter 10, “Command-Line Beginner’s Class,” but have only glossed over the idea, calling it a command line and terminal. Simply put, a *shell* is a program that users employ to type commands. In our case, it is an interpreter that supplies both a command language and a program scripting language that can control what the operating system does.

Several shells are available for users. The most widely known and widely distributed shells early in the UNIX

IN THIS CHAPTER

- ▶ What Is a Shell?
- ▶ Scheduling Tasks
- ▶ Basic Shell Control
- ▶ Writing and Executing a Shell Script
- ▶ References

era were the Bourne Shell, `sh`, and the C Shell, `csh`. The C Shell used syntax from the C programming language whereas the Bourne Shell developed a unique set of features that today have become standard to all UNIX/Linux shells.

Here is a list of the most commonly found shells these days:

- ▶ **Almquist Shell (`ash`):** This was written as a BSD-licensed replacement for the Bourne Shell, with just slightly fewer features (mostly eliminating those not frequently used).
- ▶ **Bourne-Again Shell (`bash`):** This was written by the GNU Project as a GPL-licensed version of the Bourne Shell with some enhancements.
- ▶ **Debian Almquist Shell (`dash`):** This is written for use in resource-constrained environments and is considered a modern replacement for the Almquist Shell.
- ▶ **Korn Shell (`ksh` or `pksh`):** This was written by David Korn while he worked at Bell Labs and is based on the Bourne Shell source code. It has been released under the Eclipse Public License; the `pksh` version is the one you are most likely to encounter as it is released with a public domain license.
- ▶ **TENEX C Shell (`tcsh`):** This is based on and compatible with the C Shell, but released with a BSD license.
- ▶ **Z Shell (`zsh`):** This is a modern Bourne-Again Shell replacement with some improvements and is the default shell in macOS.

The automation and tasks described in this chapter should work on most shells.

By default, Ubuntu uses the Bourne-Again Shell (`bash`). It is also aliased to `/bin/sh`, so whenever you run a shell script using `sh` notation, you are using `bash` unless you have chosen to use a different shell.

To temporarily change your shell, just for your current session, enter the name of the shell you want to use. If it is not yet installed, most of them are available in the Ubuntu software repositories via `apt`.

To permanently change your shell, you use an interactive program, `chsh`, which stands for “change shell.”

Scheduling Tasks

There are three ways to schedule commands in Ubuntu, all of which work in different ways. The first is the `at` command, which specifies a command to run at a specific time and date relative to today. The second is the `batch` command, which is actually a script that redirects you to the `at` command with some extra options set so your command runs when the system is quiet. The last option is the `cron` daemon, which is the Linux way of executing tasks at a given time.

Using `at` and `batch` to Schedule Tasks for Later

If you want to run a time-intensive task, but you do not want to do it while you are logged in, you can tell Ubuntu to run it later with the `at` command, which you must install. The package name is the same as the tool: `at`. To use `at`, you need to tell it the time at which you want the task to run and then press Enter. You then see a new prompt that starts with `at>`, and everything you type there until you press Ctrl+D will be the commands that `at` will run.

When the designated time arrives, `at` performs each action individually and in order, which means later commands can rely on the results of earlier commands. In the following example, run `at` just after 8:00 p.m., `at` is used to download and extract the latest Linux kernel at a time when the network should be quiet:

```
matthew@seymour:~$ at now + 7 hours
at> wget http://www.kernel.org/pub/linux/kernel/v3.0/linux-3.0.tar.bz2
at> tar xvfjp linux-3.0.tar.bz2
at> <EOT>
job 2 at 2011-07-08 20:01
```

Specifying `now + 7 hours` as the time does what you would expect: `at` was run at 8:00 p.m., so the command will run just after 3:00 a.m.

If you have a more complex job, you can use the `-f` parameter to have `at` read its commands from a file, like this:

```
echo wget http://www.kernel.org/pub/linux/kernel/v3.0/linux-3.00.tar.bz2\;
tar xvfjp linux-3.0.tar.bz2 > myjob.job
at -f myjob.job tomorrow
```

As you can see, `at` is flexible about the time format it takes; you can specify it in three ways:

- ▶ Using the `now` parameter, you can specify a number of minutes, hours, days, or weeks relative to the current time. For example, `now + 4 weeks` runs the command four weeks from today.
- ▶ You can also specify several special times, including `tomorrow`, `midnight`, `noon`, or `teatime` (4:00 p.m.). If you do not specify a time with `tomorrow`, your job is set for precisely 24 hours from the current time.
- ▶ You can specify an exact date and time by using `HH:MM MM/DD/YY` format (for example, `16:40 22/12/20` for 4:40 p.m. on December 22, 2020).

When your job is submitted, `at` reports the job number, date, and time that the job will be executed; the queue identifier; and the job owner (you). It also captures all your environment variables and stores them along with the job so that when your job runs, it can restore the variables, preserving your execution environment.

The job number and job queue identifier are both important. When you schedule a job using `at`, it is placed into queue `a` by default, which means it runs at your specified time and takes up a normal amount of resources.

An alternative command, `batch`, is really just a shell script that calls `at` with a few extra options. These options (`-q b -m now`, in case you are interested) set `at` to run on queue `b` (`-q b`), mail the user on completion (`-m`), and run immediately (`now`). The queue part is what is important: Jobs scheduled on queue `b` will only be executed when system load falls below 0.8—that is, when the system is not running at full load. Furthermore, those jobs run with a lower niceness, meaning a queue jobs usually have a niceness of 2, whereas queue `b` jobs have a niceness of 4.

Because `batch` always specifies `now` as its time, you need not specify your own time; the task will simply run as soon as the system is quiet. Having a default niceness of 4 means that batched commands get fewer system resources than a queue job gets (`at`'s default) and fewer system resources than with most other programs. You can optionally specify other queues by using `at`. Queue `c` runs at niceness 6, queue `d` runs at niceness 8, and so on. However, it is important to note that the system load is only checked before the command is run. If the load is lower than 0.8, your batch job runs. If the system load subsequently rises beyond 0.8, your batch job continues to run, albeit in the background, thanks to its niceness value.

When you submit a job for execution, you are also returned a job number. If you forget this or just want to see a list of other jobs you have scheduled to run later, use the `atq` command with no parameters. If you run this as a normal user, it prints only your jobs; running it as a super user prints everyone's jobs. The output is in the same format as when you submit a job, so you get the ID number, execution time, queue ID, and owner of each job.

If you want to delete a job, use the `atrm` command followed by the ID number of the job you want to delete. This next example shows `atq` and `atrm` being used to list jobs and delete one:

```
matthew@seymour:~$ atq
14      2012-01-20 23:33 a matthew
16      2012-02-03 22:34 a matthew
17      2012-01-25 22:34 a matthew
15      2012-01-22 04:34 a matthew
18      2012-01-22 01:35 b matthew
matthew@seymour:~$ atrm 16
matthew@seymour:~$ atq
14      2012-01-20 23:33 a matthew
17      2012-01-25 22:34 a matthew
15      2012-01-22 04:34 a matthew
18      2012-01-22 01:35 b matthew
```

In this example, job 16 is deleted using `atrm`, and so it does not show up in the second call to `atq`.

The default configuration for `at` and `batch` is to allow everyone to use it, which is not always the desired behavior. Access is controlled through two files: `/etc/at.allow` and `/etc/at.deny`. By default, `at.deny` exists but is empty, which allows everyone to use `at` and `batch`. You can enter usernames into `at.deny`, one per line, to stop those users from scheduling jobs.

Alternatively, you can use the `at.allow` file; this does not exist by default. If you have a blank `at.allow` file, no one except `root` is allowed to schedule jobs. As with `at.deny`, you can add usernames to `at.allow`, one per line, and those users are able to schedule jobs. You should use either `at.deny` or `at.allow`: When someone tries to run `at` or `batch`, Ubuntu checks for her username in `at.allow`. If it is in there, or if `at.allow` does not exist, Ubuntu checks for her username in `at.deny`. If her username is in `at.deny` or if `at.deny` does not exist, the user is not allowed to schedule jobs.

Using `cron` to Run Jobs Repeatedly

The `at` and `batch` commands work well if you just want to execute a single task at a later date, but they are less useful if you want to run a task frequently. Instead, the `cron` daemon exists for running tasks repeatedly based on system (and user) requests. The `cron` daemon has a similar permissions system to `at`: Users listed in the `cron.deny` file are not allowed to use `cron`, and users listed in the `cron.allow` file are. An empty `cron.deny` file—the default—means everyone can set jobs. An empty `cron.allow` file means that no one (except `root`) can set jobs.

There are two types of jobs: system jobs and user jobs. Only `root` can edit *system* jobs, whereas any user whose name appears in `cron.allow` or does not appear in `cron.deny` can run *user* jobs. System jobs are controlled through the `/etc/crontab` file, which by default looks like this:

```
SHELL=/bin/sh
PATH=/usr/local/sbin:/usr/local/bin:/sbin:/bin:/usr/sbin:/usr/bin

# m h dom mon dow user  command
17 * * * * root    cd / && run-parts -report /etc/cron.hourly
25 6 * * * root    test -x /usr/sbin/anacron || ( cd / && run-parts -report
[ccc]/etc/cron.daily )
47 6 * * 7 root    test -x /usr/sbin/anacron || ( cd / && run-parts -report
[ccc]/etc/cron.weekly )
52 6 1 * * root    test -x /usr/sbin/anacron || ( cd / && run-parts -report
[ccc]/etc/cron.monthly )
```

The first two lines specify which shell should be used to execute the job (it defaults to the shell of the user who owns the `crontab` file, usually `/bin/bash`) and the search path for executables that will be used. It's important that you avoid using environment variables in this path statement because they might not be set when the job runs.

The next line starts with a pound sign (`#`) and so is treated as a comment and ignored. The next four lines are the important parts: They are the jobs themselves.

Each job is specified in seven fields that define the time to run, owner, and command. The first five commands specify the execution time in quite a quirky order: minute (0–59), hour (0–23), day of the month (1–31), month of the year (1–12), and day of the week (0–7). For day of the week, both 0 and 7 are Sunday, which means that 1 is Monday, 3 is Wednesday, and so on. If you want to specify “all values” (that is, every minute, every hour, every day, and so on), use an asterisk, `*`.

The next field specifies the username of the owner of the job. When a job is executed, it uses the username specified here. The last field is the command to execute.

So, the first job runs at minute 17, every hour of every day of every month, and executes the command `run-parts /etc/cron.hourly`. The `run-parts` command is a simple script that runs all programs inside a given directory (in this case, `/etc/cron.hourly`). So, in this case, the job executes at 00:17 (17 minutes past midnight), 01:17, 02:17, 03:17, and so on and uses all the programs listed in the `cron.hourly` directory.

The next job runs at minute 25 and hour 6 of every day of every month, running `run-parts /etc/cron.daily`. Because of the hour limitation, this script runs only once per day, at 6:25 a.m. Note that it uses minute 25 rather than minute 17 so that daily jobs do not clash with hourly jobs. You should be able to guess what the next two jobs do simply by looking at the commands they run.

Each of those four directories (`cron.hourly`, `cron.daily`, `cron.weekly`, and `cron.monthly`) contains a collection of shell scripts that are run by `run-parts`. For example, in `cron.daily` you have scripts like `logrotate`, which handles backing up of log files, and `make-whatis`, which updates the `whatis` database. You can add other system tasks to these directories if you want to, but be careful to ensure that your scripts are correct.

CAUTION

The `cron` daemon reads all the system `crontab` files and all user `crontab` files once a minute (on the minute; that is, at 6:00:00, 6:01:00, and so on) to check for changes. However, any new jobs it finds will not be executed until at least 1 minute has passed.

For example, if it is 6:01:49 (that is, 49 seconds past 1 minute past 6:00 a.m.) and you set a `cron` job to run at 6:02, it does not execute. At 6:02, the `cron` daemon rereads its configuration files and sees the new job, but it is not able to execute it. If you set the job to run at 6:02 a.m. every day, it is executed the following morning and every subsequent morning.

This same situation exists when deleting jobs. If it is 6:01:49 and you have a job scheduled to run at 6:02, deleting it makes no difference: `cron` runs it before it rereads the `crontab` files for changes. However, after it has reread the `crontab` file and noticed that the job is no longer there, it will not be executed in subsequent days.

There are alternative ways of specifying dates. For example, you can use sets of dates and times by using hyphens or commas; for example, hours 9–15 would execute at 9, 10, 11, 12, 13, 14, and 15 (from 9:00 a.m. to 3:00 p.m.), whereas 9, 11, 13, 15 would skip the even hours. Note that it is important that you not put spaces into these sets because the `cron` daemon interprets a space as the next field. You can define a step value with a slash

(/) to show time division; for example, */4 for hours means “every 4 hours all day,” and 0-12/3 means “every 3 hours from midnight to noon.” You can also specify day and month names rather than numbers, using three-character abbreviations: Sun, Mon, Tue, Fri, Sat for days or Jan, Feb, Mar, Oct, Nov, Dec for months.

As well as system jobs, there are user jobs for those users who have the correct permissions. User jobs are stored in the `/var/spool/cron` directory, with each user having his own file named after his username (for instance, `/var/spool/cron/philip` or `/var/spool/cron/root`). These files contain the jobs the user wants to run and take roughly the same format as the `/etc/crontab` file, with the exception that the owner of the job should not be specified because it is always the same as the filename.

To edit your own `crontab` file, type `crontab -e`. This brings up a text editor, where you can enter your entries. (By default the editor is `vim`, also known by its older name `vi`, but you can set the `EDITOR` environment variable to change that.) The format of this file is a little different from the format for the main `crontab` file because in this case there is no need to specify the owner of the job, as it is always you.

So, this time each line is made up of six fields: minute (0–59), hour (0–23), day of the month (1–31), month of the year (1–12), day of the week (0–7), and the command to run. If you are using `vim` and are new to it, press `i` to enter insert mode to edit your text; then press `Esc` to exit insert mode. To save and quit, type a colon followed by `wq` and press `Enter`.

When programming, we tend to use a `sandbox` subdirectory in our home directory where we keep all sorts of temporary files that we were just playing around with. We can use a personal job to empty that directory every morning at 6:00 a.m. to get a fresh start each morning. Here is how that would look in our `crontab` file:

```
0 6 * * * rm -rf /home/matthew/sandbox/*
```

If you are not allowed to schedule jobs, you will be stopped from editing your `crontab` file.

After your jobs are placed, you can use the command `crontab -l` to list your jobs. This just prints the contents of your `crontab` file, so its output is the same as the line you just entered.

If you want to remove just one job, the easiest thing to do is type `crontab -e` to edit your `crontab` file in `vim`; then, after having moved the cursor to the job you want to delete, type `dd` to delete that line. If you want to delete all your jobs, you can use `crontab -r` to delete your `crontab` file.

Read the man page for more about `cron`.

Using `rtcwake` to Wake Your Computer from Sleep Automatically

Some of us keep our computers running 24/7. Perhaps you don't want to do so, and you need to have your system up and running at a certain time every day, but you can't guarantee that you will be able to be present to turn it on. It is possible to use `rtcwake` to place

the computer in sleep or suspend mode instead of turning it off and then wake up the computer later. To do this, you must have `sudo` permissions. Here is an example:

```
matthew@seymour:~$ sudo rtcwake -m mem -s -3600
```

This command tells the computer to suspend to RAM, or sleep, which means to save the current state of the computer in memory and shut down everything else and then to wake the computer after 3600 seconds, which is one hour.

Here is the basic syntax of the command:

```
sudo rtcwake -m [type of suspend] -s [number of seconds]
```

There are five types of suspend available to use with `-m`:

- ▶ **disk**—(hibernate) The current state of the computer is written to disk, and the computer is powered off.
- ▶ **mem**—(sleep) The current state of the computer is written to RAM, and the computer is put into a low-power state, using just enough power to keep the memory preserved.
- ▶ **no**—The computer is not suspended immediately. Only the wakeup time is set. This allows you to continue working; you have to remember to put the computer to sleep manually.
- ▶ **off**—The computer is turned off completely. Wake will not work with this setting for everyone and is not officially supported, but it does work with some computers. It is included here for those who like to live dangerously.
- ▶ **standby**—The computer is put into standby mode, which saves some power over running normally but not nearly as much as the other options. This is the default setting and will be used if you omit `-m`.

Setting the wake time can be done more than one way:

- ▶ As shown earlier, you can use `-s`, which specifies the number of seconds before waking.
- ▶ You can also use `-t`, which allows you to set a specific time to wake but formatted in the number of seconds since the beginning of UNIX time (00:00:00 UTC on 1/1/1970). The `date` command can help you find this number, which is a commonly used method of performing time-related tasks in the UNIX/Linux world. You can do so like this: `sudo rtcwake -m no -t $(date +%s -d 'tomorrow 06:30')`.

See the man files for `rtcwake` and `date` for help and more options.

Here are a few tips to help you get started:

- ▶ The letters *RTC* stand for “real-time clock,” which refers to the hardware clock that is set in your BIOS and is kept running by the battery on your motherboard. If your

computer needs a new battery, as evidenced by the time needing to be reset every time you turn the computer back on, or if you have other clock-related problems, this command will not work for you.

- ▶ If you have problems using `sleep`, `hibernate`, or `suspend` on your system, this command will not work for you.
- ▶ You probably want to avoid using this command on a notebook computer. Overheating and/or dead batteries are a real possibility if a system wakes itself while the computer is in a laptop bag.
- ▶ If you want to run a specific command when the computer wakes up, you can do this the same way you chain other commands to run in a series: Put `&&` after `rtc-wake` and before the command you want to run when `rtcwake` has completed, as discussed in Chapter 12, “Command-Line Master Class, Part 2.”

Basic Shell Control

Ubuntu includes a rich assortment of capable, flexible, and powerful shells. Each shell is different but has numerous built-in commands and configurable command-line prompts and might include features such as command-line history, the ability to recall and use a previous command line, and command-line editing. As an example, the `bash` shell is so powerful that it is possible to write a minimal web server entirely in `bash`'s language using 114 lines of script. (See the link for the `bash` home page at the end of this chapter.)

The Shell Command Line

Having a basic understanding of the capabilities of the shell command line can help you write better shell scripts. If, after you have finished reading this short introduction, you want to learn more about the command line, check out Chapter 11, “Command-Line Master Class, Part 1.” You can use the shell command line to perform a number of different tasks, including the following:

- ▶ Searching files or directories with programs using pattern matching or expressions. These commands include the GNU `gawk` (linked as `awk`) and the `grep` family of commands, including `egrep` and `fgrep`.
- ▶ Getting data from and sending data to a file or command, known as *input and output redirection*.
- ▶ Feeding or filtering a program's output to another command (called using *pipes*).

A shell can also have built-in *job-control* commands to launch the command line as a background process, suspend a running program, selectively retrieve or kill running or suspended programs, and perform other types of process control.

You can run multiple commands on a single command line by using a semicolon to separate commands:

```
matthew@seymour:~$ w ; free ; df
 18:14:13 up  4:35,  2 users,  load average: 0.97, 0.99, 1.04
USER      TTY      FROM          LOGIN@      IDLE        JCPU   PCPU WHAT
matthew   tty7     :0            13:39       4:35m 24:34   0.32s gnome-session
matthew   pts/0    :0.0          17:24       0.00s  1.19s  4.98s gnome-terminal
          total    used      free      shared    buffers  cached
Mem:      4055692 1801104   2254588      0         134096   757532
-/+ buffers/cache:  909476 3146216
Swap:     8787512 0         8787512
Filesystem      1K-blocks    Used Available Use% Mounted on
/dev/sda1       14421344    6509276   7179508  48% /
none            2020136      336    2019800   1% /dev
none            2027844      3004   2024840   1% /dev/shm
none            2027844       224   2027620   1% /var/run
none            2027844        0   2027844   0% /var/lock
none            2027844        0   2027844   0% /lib/init/rw
/dev/sda6       284593052 144336704 125799860  54% /home
```

This example displays the output of the `w`, `free`, and `df` commands. You can extend long shell command lines inside shell scripts or at the command line by using the backslash character (`\`), as follows:

```
matthew@seymour:~$ echo "this is a long \
> command line and" ; echo "shows that multiple commands \
> may be strung out."
this is a long command line and
shows that multiple commands may be strung out.
```

The first three lines of this example are a single command line. In that single line are two instances of the `echo` command. Note that when you use the backslash as a line-continuation character, it must be the last character on the command line (or in your shell script, as you see later in this chapter, in the “Writing and Executing a Shell Script” section).

Using the basic features of the shell command line is easy, but mastering the use of all features can be difficult. Entire books have been devoted to using shells, writing shell scripts, and using pattern-matching expressions. The following sections provide an overview of some features of the shell command line related to writing scripts.

UNDERSTANDING `grep`

If you plan to develop shell scripts to expand the capabilities of pattern-matching commands such as `grep`, you will benefit from learning more about using expressions. One of the definitive guides to using the pattern-matching capabilities of UNIX and Linux commands is *Mastering Regular Expressions* by Jeffrey E. F. Friedl.

Shell Pattern-Matching Support

The shell command line enables you to use strings of specially constructed character patterns for wildcard matches. This is a different, simpler capability than that supported by GNU utilities such as `grep`, which can use more complex patterns, known as *expressions*, to search through files or directories or to filter data input to or out of commands.

The shell's pattern strings can be simple or complex, but even using a small subset of the available characters in simple wildcards can yield constructive results at the command line. Common characters used for shell pattern matching include the following:

- ▶ `*`—Matches any character. For example, to find all files in the current directory ending in `.txt`, you could use this:

```
matthew@seymour:~$ ls *.txt
```

- ▶ `?`—Matches a single character. For example, to find all files in the current directory ending in the extension `.d?c` (where `?` could be 0–9, a–z, or A–Z), you could use the following:

```
matthew@seymour:~$ ls *.d?c
```

- ▶ `[xxx]` or `[x-x]`—Matches a range of characters. For example, to list all files in a directory with names containing numbers, you could use this:

```
matthew@seymour:~$ ls *[0-9]*
```

- ▶ `\x`—Matches or escapes a character such as `?` or a tab character. For example, to create a file with a name containing a question mark, you could use the following:

```
matthew~$ touch foo\?
```

Note that the shell might not interpret some characters or regular expressions in the same manner as a Linux command, and mixing wildcards and regular expressions in shell scripts can lead to problems unless you're careful. For example, finding patterns in text is best left to regular expressions used with commands such as `grep`; simple wildcards should be used for filtering or matching filenames on the command line. And although both Linux command expressions and shell scripts can recognize the backslash as an escape character in patterns, the dollar sign (`$`) has two wildly different meanings (single-character pattern matching in expressions and variable assignment in scripts).

CAUTION

Make sure you read your command carefully when using wildcards; an all-too-common error is to type something like `rm -rf * .txt` with a space between the `*` and `.txt`. By the time you wonder why the command is taking so long, Bash will already have deleted most of your files. The problem is that it will treat the `*` and the `.txt` separately. `*` will match everything, so Bash will delete all your files.

Redirecting Input and Output

You can create, overwrite, and append data to files at the command line, using a process called *input and output redirection*. The shell recognizes several special characters for this process, such as `>`, `<`, or `>>`.

In this example, the output of the `ls` command is redirected to create a file named `textfiles.listing`:

```
matthew@seymour:~$ ls *.txt >textfiles.listing
```

Use output redirection with care because it is possible to overwrite existing files. For example, specifying a different directory but using the same output filename overwrites the existing `textfiles.listing`:

```
matthew@seymour:~$ ls /usr/share/doc/mutt-1.4/*.txt >textfiles.listing
```

Fortunately, most shells are smart enough to recognize when you might do something foolish. Here, the `bash` shell warns that the command is attempting to redirect output to a directory:

```
matthew@seymour:~$ mkdir foo
matthew@seymour:~$ ls >foo
bash: foo: Is a directory
```

Output can be appended to a file without overwriting existing content by using the append operator, `>>`. In this example, the directory listing is appended to the end of `textfiles.listing` instead of overwriting its contents:

```
matthew@seymour:~$ ls /usr/share/doc/mutt-1.4/*.txt >>textfiles.listing
```

You can use *input redirection* to feed data into a command by using the `<` like this:

```
matthew@seymour:~$ cat < textfiles.listing
```

You can use the shell *here* operator, `<<`, to specify the end of input on the shell command line:

```
matthew@seymour:~$ cat >simple_script <<DONE
> echo ""this is a simple script""
> DONE
matthew@seymour:~$ cat simple_script
echo ""this is a simple script""
```

In this example, the shell feeds the `cat` command you are typing (input) until the pattern `DONE` is recognized. The output file `simple_script` is then saved and its contents verified. You can use this same technique in scripts to create content based on the output of various commands and define an end-of-input or delimiter.

Piping Data

Many Linux commands can be used in concert in a single, connected command line to transform data from one form to another. Stringing together Linux commands in this fashion is known as using or creating *pipes*. Pipes are created on the command line with the bar operator (`|`). For example, you can use a pipe to perform a complex task from a single command line like this:

```
matthew@seymour:~$ find /d2 -name '*.txt' -print | xargs cat | \
tr ' ' '\n' | sort | uniq >output.txt
```

This example takes the output of the `find` command to feed the `cat` command (via `xargs`) the names of all text files in the `/d2` directory. The content of all matching files is then fed through the `tr` command to change each space in the data stream into a carriage return. The stream of words is then sorted, and identical adjacent lines are removed using the `uniq` command. The output, a raw list of words, is then saved in the file named `output.txt`.

Background Processing

The shell allows you to start a command and then launch it into the background as a process by using an ampersand (`&`) at the end of a command line. This technique is often used at the command line of an X terminal window to start a client and return to the command line. For example, to launch another terminal window using the `xterm` client, you can use the following:

```
matthew@seymour:~$ xterm &
[3] 1437
```

The numbers echoed back show a number (3 in this example), which is a *job* number, or reference number for a shell process, and a *process ID* number, or PID (1437 in this example). You can kill the `xterm` window session by using the shell's built-in `kill` command, along with the job number, like this:

```
matthew@seymour:~$ kill %3
```

Or you can kill the process by using the `kill` command, along with the PID, as follows:

```
matthew@seymour:~$ kill 1437
```

You can use background processing in shell scripts to start commands that take a long time, such as backups:

```
matthew@seymour:~$ tar -czf /backup/home.tgz /home &
```

Writing and Executing a Shell Script

Why should you write and use shell scripts? Shell scripts can save you time and typing, especially if you routinely use the same command lines multiple times every day.

Although you could also use the history function (by pressing the up or down arrow keys while using `bash` or using the `history` command), a shell script can add flexibility with command-line argument substitution and built-in help.

Although a shell script doesn't execute faster than a program written in a computer language such as C, a shell program can be smaller in size than a compiled program. The shell program does not require any additional library support other than the shell or, if used, existing commands installed on your system. The process of creating and testing shell scripts is also generally simpler and faster than the development process for equivalent C language commands.

NOTE

Hundreds of commands included with Ubuntu are actually shell scripts, and many other good shell script examples are available over the Internet; a quick search yields numerous links to online tutorials and scripting guides from fellow Linux users and developers. For example, the `startx` command, used to start an X Window session from the text console, is a shell script used every day by most users. To learn more about shell scripting with `bash`, see the "Advanced Bash-Scripting Guide," listed in the "References" section at the end of this chapter. You'll also find *Teach Yourself Shell Programming in 24 Hours* a helpful guide to learning more about using the shell to build your own commands.

When you are learning to write and execute your first shell scripts, start with scripts for simple but useful tasks. Begin with short examples and then expand the scripts as you build on your experience and knowledge. Make liberal use of comments (lines preceded with a pound sign, #) to document each section of your script. Include an author statement and an overview of the script as additional help, along with a creation date or version number. Write shell scripts using a text editor such as `vi` because it does not automatically wrap lines of text. Line wrapping can break script syntax and cause problems. If you use the `nano` editor, include its `-w` flag to disable line wrapping.

In this section, you learn how to write a simple shell script to set up a number of *aliases* (command synonyms) whenever you log on. Instead of typing all the aliases every time you log on, you can put them in a file by using a text editor, such as `vi`, and then execute the file. Normally these changes are saved in system-wide shell configuration files under the `/etc` directory to make the changes active for all users or in your `.bashrc`, `.cshrc` (if you use `tcsh`), or `.bash_profile` files in your home directory.

Here is what is contained in `myenv`, a sample shell script created for this purpose (for `bash`):

```
#!/bin/sh
alias ll='ls -L'
alias ldir='ls -aF'
alias copy='cp'
```

This simple script creates command *aliases*, or convenient shorthand forms of commands, for the `ls` and `cp` commands. The `ll` alias provides a long directory listing: The `ldir` alias is the `ls` command but prints indicators (for directories or executable files) in listings.

The `copy` alias is the same as the `cp` command. You can experiment and add your own options or create aliases of other commands with options you frequently use.

You can execute `myenv` in a variety of ways under Linux. As shown in this example, you can make `myenv` executable by using the `chmod` command and then execute it as you would any other native Linux command:

```
matthew@seymour:~$ chmod +x myenv
```

This line turns on the executable permission of `myenv`, which can be checked with the `ls` command and its `-l` option like this:

```
matthew@seymour:~$ ls -l myenv
-rwxr-xr-x  1 matthew matthew  0 2010-07-08 18:19 myenv
```

Running the New Shell Program

You can run your new shell program in several ways. Each method produces the same results, which is a testament to the flexibility of using the shell with Linux. One way to run your shell program is to execute the file `myenv` from the command line as if it were a Linux command:

```
matthew@seymour:~$ ./myenv
```

A second way to execute `myenv` under a particular shell, such as `pdksh`, is as follows:

```
matthew@seymour:~$ pdksh myenv
```

This invokes a new `pdksh` shell and passes the filename `myenv` as a parameter to execute the file. A third way requires you to create a directory named `bin` in your home directory and to then copy the new shell program into this directory. You can then run the program without the need to specify a specific location or to use a shell. You do so like this:

```
matthew@seymour:~$ mkdir bin
matthew@seymour:~$ mv myenv bin
matthew@seymour:~$ myenv
```

This works because Ubuntu is set up by default to include the executable path `$HOME/bin` in your shell's environment. You can view this environment variable, named `PATH`, by piping the output of the `env` command through `fgrep`, like so:

```
matthew@seymour:~$ env | fgrep PATH
/usr/kerberos/bin:/usr/local/bin:/bin:/usr/bin: \
/usr/X11R6/bin:/sbin:/home/matthew/bin
```

As you can see, the user (`matthew` in this example) can use the new `bin` directory to hold executable files. Another way to bring up an environment variable is to use the `echo` command along with the variable name (in this case, `$PATH`):

```
matthew@seymour:~$ echo $PATH
/usr/kerberos/bin:/usr/local/bin:/usr/bin:/bin:/usr/X11R6/bin:/home/bball/bin
```

CAUTION

Never put `.` in your `$PATH` to execute files or a command in the current directory; doing so presents a serious security risk, especially for the root operator, and even more so if `.` is first in your `$PATH` search order. Trojan scripts placed by crackers in directories such as `/tmp` can be used for malicious purposes and will be executed immediately if the current working directory is part of your `$PATH`.

Storing Shell Scripts for System-wide Access

After you execute the command `myenv`, you should be able to use `ldir` from the command line to get a list of files under the current directory and `ll` to get a list of files with attributes displayed. However, the best way to use the new commands in `myenv` is to put them into your shell's login or profile file. For Ubuntu users, and nearly all Linux users, the default shell is `bash`, so you can make these commands available for everyone on your system by putting them in the `/etc/bashrc` file. System-wide aliases for `tcsh` are contained in files with the extension `.csh` under the `/etc/profile.d` directory. The `pksh` shell can use these command aliases, as well.

NOTE

To use a shell other than `bash` after logging in, use the `chsh` command from the command line or the `system-config-users` client during an X session. You'll be asked for your password (or the root password, if using `system-config-users`) and the location and name of the new shell. The new shell will become your default shell, but only if its name is in the list of acceptable system shells in `/etc/shells`.

Interpreting Shell Scripts Through Specific Shells

The majority of shell scripts use a *shebang line* (`#!`) at the beginning to control the type of shell used to run the script; this bang line calls for an `sh`-incantation of `bash`:

```
#!/bin/sh
```

A shebang line (it is short for “sharp” and “bang,” two names for `#` and `!`) tells the Linux kernel that a specific command (a shell, or in the case of other scripts, perhaps `awk` or Perl) is to be used to interpret the contents of the file. Using a shebang line is common practice for all shell scripting. For example, if you write a shell script using `bash` but want the script to execute as if run by the Bourne shell, `sh`, the first line of your script contains `#!/bin/sh`, which is a link to the `dash` shell. Running `dash` as `sh` causes `dash` to act as a Debian Almquist shell. This is the reason for the symbolic link `sh`, which points to `dash`.

THE SHEBANG LINE

The shebang line is a magic number, as defined in `/usr/share/misc/magic`—a text database of magic numbers for the Linux `file` command. Magic numbers are used by many different Linux commands to quickly identify a type of file, and the database format is

documented in the section five manual page named `magic` (read by using `man 5 magic`). For example, magic numbers can be used by the Linux `file` command to display the identity of a script (no matter what filename is used) as a shell script using a specific shell or other interpreter, such as `awk` or `Perl`.

You might also find different or new environmental variables available to your scripts by using different shells. For example, if you launch `csH` from the `bash` command line, you find several new variables or variables with slightly different definitions, such as the following:

```
matthew@seymour:~$ env
...
VENDOR=intel
MACHTYPE=i386
HOSTTYPE=i386-linux
HOST=thinkpad.home.org
```

On the other hand, `bash` might provide these variables or variables of the same name with a slightly different definition, such as the following:

```
matthew@seymour:~$ env
...
HOSTTYPE=i386
HOSTNAME=thinkpad.home.org
```

Although the behavior of a shebang line is not defined by POSIX, variations of its use can prove helpful when you are writing shell scripts. For example, as described in the `wish` man page, you can use a shell to help execute programs called within a shell script without needing to hardcode pathnames of programs. The `wish` command is a windowing *Tool Control Language* (`tc1`) interpreter that can be used to write graphical clients. Avoiding the use of specific pathnames to programs increases shell script portability because not every UNIX or Linux system has programs in the same location.

For example, if you want to use the `wish` command, your first inclination might be to write this:

```
#!/usr/local/bin/wish
```

Although this works on many other operating systems, the script fails under Linux because `wish` is located in the `/usr/bin` directory. However, if you write the command line this way, you can use the `wish` command (as a binary or a shell script itself):

```
#!/bin/sh
exec wish "$@"
```


Using Variables in Shell Scripts

When writing shell scripts for Linux, you work with three types of variables:

- ▶ **Environment variables**—You can use these variables, which are part of the system environment, in your shell program. You can define new variables, and you can also modify some of them, such as `PATH`, within a shell program.
- ▶ **Built-in variables**—Variables such as options used on the command (interpreted by the shell as a *positional argument*) are provided by Linux. Unlike environment variables, you cannot modify built-in variables.
- ▶ **User variables**—These variables are defined within a script when you write a shell script. You can use and modify them at will within the shell script, but they are not available to be used outside the script.

A major difference between shell programming and other programming languages is that in shell programming, variables are not *typed*—that is, you do not have to specify whether a variable is a number or a string, and so on.

Assigning a Value to a Variable

Suppose that you want to use a variable called `lcount` to count the number of iterations in a loop within a shell program. You can declare and initialize this variable as follows:

Command	Environment
<code>lcount=0</code>	<code>pdksh</code> and <code>bash</code>
<code>set lcount=0</code>	<code>tcsh</code>

NOTE

Under `pdksh` and `bash`, you must ensure that the equals sign (=) does not have spaces before and after it.

To store a string in a variable, you can use the following:

Command	Environment
<code>myname=Sedona</code>	<code>pdksh</code> and <code>bash</code>
<code>set myname=Sedona</code>	<code>tcsh</code>

Use the preceding variable form if the string doesn't have embedded spaces. If a string has embedded spaces, you can do the assignment as follows:

Command	Environment
<code>myname="Sage"</code>	<code>pdksh</code> and <code>bash</code>
<code>set myname="Sage"</code>	<code>tcsh</code>

Accessing Variable Values

You can access the value of a variable by prefixing the variable name with a dollar sign (\$). That is, if the variable name is `var`, you can access the variable by using `$var`.

If you want to assign the value of `var` to the variable `lcount`, you can do so as follows:

Command	Environment
<code>lcount=\$var</code>	<code>pdksh</code> and <code>bash</code>
<code>set lcount=\$var</code>	<code>tcsh</code>

Positional Parameters

Passing options from the command line or from another shell script to your shell program is possible.

These options are supplied to the shell program by Linux as *positional parameters*, which have special names provided by the system. The first parameter is stored in a variable called `1` (number 1) and can be accessed by using `$1` within the program. The second parameter is stored in a variable called `2` and can be accessed by using `$2` within the program, and so on. One or more of the higher-numbered positional parameters can be omitted while you're invoking a shell program.

Understanding how to use these positional parameters and how to access and use variables retrieved from the command line is necessary when developing more advanced shell programs.

A Simple Example of a Positional Parameter

Consider this example: If a shell program `mypgm` expects two parameters—such as a first name and a last name—you can invoke the shell program with only one parameter, the first name. However, you cannot invoke it with only the second parameter, the last name.

Here is a shell program called `mypgm1`, which takes only one parameter (a name) and displays it on the screen:

```
#!/bin/sh
#Name display program
if [ $# -eq 0 ]
then
    echo "Name not provided"
else
    echo "Your name is "$1
fi
```

If you execute `mypgm1`, as follows:

```
matthew@seymour:~$ bash mypgm1
```

you get the following output:

```
Name not provided
```

However, if you execute `mypgm1` as follows:

```
matthew@seymour:~$ bash mypgm1 Sandra
```

you get this output:

```
Your name is Sandra
```

The shell program `mypgm1` also illustrates another aspect of shell programming: the built-in variables provided to the shell by the Linux kernel. In `mypgm1`, the built-in variable `$#` provides the number of positional parameters passed to the shell program. You learn more about working with built-in variables in the next major section of this chapter.

Using Positional Parameters to Access and Retrieve Variables from the Command Line

Using positional parameters in scripts can be helpful if you need to use command lines with piped commands requiring complex arguments. Shell programs containing positional parameters can be even more convenient if the commands are infrequently used. For example, if you use your Ubuntu system with an attached voice modem as an answering machine, you can write a script to issue a command that retrieves and plays the voice messages. The following lines convert a saved sound file (in `.rmd` or voice-phone format) and pipe the result to your system's audio device:

```
#!/bin/sh
# play voice message in /var/spool/voice/incoming
rmdtopvf /var/spool/voice/incoming/$1 | pvfspd -s 8000 | \
pvftobasic >/dev/audio
```

You can then easily play back a voice message by using this script (in this case named `pmm`):

```
matthew@seymour:~$ pmm name_of_message
```

Shell scripts that contain positional parameters are often used for automating routine and mundane jobs, such as system log report generation, file system checks, user resource accounting, printer use accounting, and other system, network, or security administration tasks.

Using a Simple Script to Automate Tasks

You could use a simple script, for example, to examine your system log for certain keywords. If the script is run via your system's scheduling table, `/etc/crontab`, it can help automate security monitoring. By combining the output capabilities of existing Linux commands with the language facilities of the shell, you can quickly build a useful script to

perform a task that normally requires a number of command lines. For example, you can create a short script, named `greplog`, like this:

```
#!/bin/sh
#   name:  greplog
#   use:   mail grep of designated log using keyword
# version: v.01 08aug02
#
#   author: bb
#
# usage:  greplog [keyword] [logpathname]
#
#   bugs:  does not check for correct number of arguments

# build report name using keyword search and date
log_report=/tmp/$1.logreport.`date +%m%d%y`

# build report header with system type, hostname, date and time
echo "===== " \
    >$log_report
echo "          S Y S T E M   O N I T O R   L O G" >>$log_report
echo uname -a >>$log_report
echo "Log report for" `hostname -f` "on" `date +%c` >>$log_report
echo "===== " \
    >>$log_report ; echo "" >>$log_report

# record log search start
echo "Search for->" $1 "starting" `date +%r` >>$log_report
echo "" >>$log_report

# get and save grep results of keyword ($1) from logfile ($2)
grep -i $1 $2 >>$log_report

# build report footer with time
echo "" >>$log_report
echo "End of" $log_report at `date +%r` >>$log_report

# mail report to root
mail -s "Log Analysis for $1" root <$log_report

# clean up and remove report
rm $log_report
exit 0
```

In this example, the script creates the variable `$log_report`, which will be the filename of the temporary report. The keyword (`$1`) and first argument on the command line are used as part of the filename, along with the current date (with perhaps a better approach being

to use \$\$ instead of the date, which will append the script's PID as a file extension). Next, the report header containing some formatted text, the output of the `uname` command, and the hostname and date are added to the report. The start of the search is then recorded, and any matches of the keyword in the log are added to the report. A footer containing the name of the report and the time is then added. The report is mailed to root with the search term as the subject of the message, and the temporary file is deleted.

You can test the script by running it manually and feeding it a keyword and a pathname to the system log, `/var/log/syslog`, like this:

```
matthew@seymour:~$ sudo greplog FAILED /var/log/syslog
```

Note that your system should be running the `syslogd` daemon. If any login failures have occurred on your system, the root operator might get an email message that looks like this:

```
Date: Sun, 23 Oct 2016 16:23:24 -0400
From: root <root@righthere.home.org>
To: root@righthere.home.org
Subject: FAILED
```

```
=====
                S Y S T E M   M O N I T O R   L O G
Linux system 4.4.0-22-generic #1 Sun Oct 9 20:21:24 EDT 2016
+GNU/Linux
Log report for righthere.home.org on Sun 23 Oct 2016 04:23:24 PM EDT
=====

Search for-> FAILED starting 04:23:24 PM

Oct 23 16:23:04 righthere login[1769]: FAILED LOGIN 3 FROM (null) FOR bball,
+Authentication failure

End of /tmp/FAILED.logreport.102303 at 04:23:24 PM
```

To further automate the process, you can include command lines using the script in another script to generate a series of searches and reports.

Built-in Variables

Built-in variables are special variables that Linux provides to the shell that you can use to make decisions within a shell program. You cannot modify the values of these variables within the shell program.

The following are some of the built-in variables:

- ▶ `$#`—The number of positional parameters passed to the shell program

- ▶ `$?`—The completion code of the last command or shell program executed within the shell program (returned value)
- ▶ `$0`—The name of the shell program
- ▶ `$*`—A single string of all arguments passed at the time of invocation of the shell program

To show these built-in variables in use, here is a sample program called `mypgm2`:

```
#!/bin/sh
#my test program
echo "Number of parameters is $#"
```

```
echo "Program name is $0"
```

```
echo "Parameters as a single string is $*"
```

If you execute `mypgm2` from the command line in `pdksh` and `bash` as follows:

```
matthew@seymour:~$ bash mypgm2 Alan Turing
```

you get the following result:

```
Number of parameters is 2
```

```
Program name is mypgm2
```

```
Parameters as a single string is Alan Turing
```

Special Characters

Some characters have special meaning to Linux shells; these characters represent commands, denote specific use for surrounding text, or provide search parameters. Special characters provide a sort of shorthand by incorporating rather complex meanings into simple characters. Table 14.1 shows some special characters.

Table 14.1 Special Shell Characters

Character	Explanation
<code>\$</code>	Indicates the beginning of a shell variable name
<code> </code>	Pipes standard output to the next command
<code>#</code>	Starts a comment
<code>&</code>	Executes a process in the background
<code>?</code>	Matches one character
<code>*</code>	Matches one or more characters
<code>></code>	Redirects output
<code><</code>	Redirects input
<code>`</code>	Indicates command substitution (the backquote or backtick—the key above the Tab key on most keyboards)
<code>>></code>	Redirects output (to append to a file)

Character	Explanation
<<	Waits until the following end-of-input string (HERE operator)
[]	Specifies a range of characters
[a-z]	Specifies all characters a through z
[a, z] or [az]	Specifies characters a or z
Space	Acts as a delimiter between two words

Special characters are very useful when you're creating shell scripts, but if you inadvertently use a special character as part of a variable name or string, your program behaves incorrectly. As you learn in later parts of this chapter, you can use one of the special characters in a string if you precede it with an *escape character* (\, or backslash) to indicate that it isn't being used as a special character and shouldn't be treated as such by the program.

A few special characters deserve special note: double quotes (") ,single quotes ('), backslash (\), and backtick (`), all of which are discussed in the following sections.

Using Double Quotes to Resolve Variables in Strings with Embedded Spaces

If a string contains embedded spaces, you can enclose the string in double quotes (") so that the shell interprets the whole string as one entity instead of as more than one.

For example, if you assigned the value `abc def` (`abc` followed by one space, followed by `def`) to a variable called `x` in a shell program as follows, you would get an error because the shell would try to execute `def` as a separate command:

Command	Environment
<code>x=abc def</code>	<code>pdksh</code> and <code>bash</code>
<code>set x = adb def</code>	<code>tcsh</code>

The shell executes the string as a single command if you surround the string in double quotes, as follows:

Command	Environment
<code>x="abc def"</code>	<code>pdksh</code> and <code>bash</code>
<code>set x="abc def"</code>	<code>tcsh</code>

The double quotes resolve all variables within the string. Here is an example for `pdksh` and `bash`:

```
var="test string"
newvar="Value of var is $var"
echo $newvar
```

Here is the same example for `tcsh`:

```
set var="test string"
set newvar="Value of var is $var"
echo $newvar
```

If you execute a shell program containing the preceding three lines, you get the following result:

```
Value of var is test string
```

Using Single Quotes to Maintain Unexpanded Variables

You can surround a string with single quotes (') to stop the shell from expanding variables and interpreting special characters. When used for the latter purpose, the single quote is an *escape character*, similar to the backslash, which you learn about in the next section. Here, you learn how to use the single quote to avoid expanding a variable in a shell script. An unexpanded variable maintains its original form in the output.

In the following examples, the double quotes from the preceding examples have been changed to single quotes:

For `pdksh` and `bash`:

```
var='test string'
newvar='Value of var is $var'
echo $newvar
```

For `tcsh`:

```
set var = 'test string'
set newvar = 'Value of var is $var'
echo $newvar
```

If you execute a shell program containing these three lines, you get the following result:

```
Value of var is $var
```

As you can see, the variable `var` maintains its original format in the results rather than being expanded.

Using the Backslash as an Escape Character

As you learned earlier, the backslash (\) serves as an escape character that stops the shell from interpreting the succeeding character as a special character. Say that you want to assign the value `$test` to a variable called `var`. If you use the following command, the shell reads the special character `$` and interprets `$test` as the value of the variable `test`. No value has been assigned to `test`; a null value is stored in `var` as follows:

Command	Environment
<code>var=\$test</code>	<code>pdksh</code> and <code>bash</code>
<code>set var=\$test</code>	<code>tcsh</code>

Unfortunately, this assignment might work for `bash` and `pdksh`, but it returns an “undefined variable” error if you use it with `tcsh`. Use the following commands to correctly store `$test` in `var`:

Command	Environment
<code>var=\\$test</code>	<code>pdksh</code> and <code>bash</code>
<code>set var = \\$test</code>	<code>tcsh</code>

The backslash before the dollar sign (`\$`) signals the shell to interpret the `$` as any other ordinary character and *not to associate any special meaning* to it. You could also use single quotes (`'`) around the `$test` variable to get the same result.

Using the Backtick to Replace a String with Output

You can use the backtick (```) character to signal the shell to replace a string with its output when executed. This is called *command substitution*. You can use this special character in shell programs when you want the result of the execution of a command to be stored in a variable. For example, if you want to count the number of lines in a file called `test.txt` in the current directory and store the result in a variable called `var`, you can use the following commands:

Command	Environment
<code>var=`wc -l test.txt`</code>	<code>pdksh</code> and <code>bash</code>
<code>set var = `wc -l test.txt`</code>	<code>tcsh</code>

Comparison of Expressions in `pdksh` and `bash`

Comparing values or evaluating the differences between similar bits of data—such as file information, character strings, or numbers—is a task known as *comparison of expressions*. Comparison of expressions is an integral part of using logic in shell programs to accomplish tasks. The way the logical comparison of two operators (numeric or string) is done varies slightly in different shells. In `pdksh` and `bash`, a command called `test` can be used to achieve comparisons of expressions. In `tcsh`, you can write an expression to accomplish the same thing.

This section covers comparison operations using the `pdksh` or `bash` shells. Later in the chapter, you learn how to compare expressions in the `tcsh` shell.

The `pdksh` and `bash` shell syntax provide a command named `test` to compare strings, numbers, and files. The syntax of the `test` command is as follows:

```
test expression
or
[ expression ]
```

Both forms of the `test` commands are processed the same way by `pdksh` and `bash`. The `test` commands support the following types of comparisons:

- ▶ String comparison
- ▶ Numeric comparison

- ▶ File operators
- ▶ Logical operators

String Comparison

You can use the following operators to compare two string expressions:

- ▶ `==`—Compares whether two strings are equal
- ▶ `!=`—Compares whether two strings are not equal
- ▶ `-n`—Evaluates whether the string length is greater than zero
- ▶ `-z`—Evaluates whether the string length is equal to zero

Next are some examples using these operators when comparing two strings, `string1` and `string2`, in a shell program called `compare1`:

```
#!/bin/sh
string1="abc"
string2="abd"
if [ $string1 = $string2 ]; then
    echo "string1 equal to string2"
else
    echo "string1 not equal to string2"
fi

if [ $string2 != string1 ]; then
    echo "string2 not equal to string1"
else
    echo "string2 equal to string2"
fi

if [ $string1 ]; then
    echo "string1 is not empty"
else
    echo "string1 is empty"
fi

if [ -n $string2 ]; then
    echo "string2 has a length greater than zero"
else
    echo "string2 has length equal to zero"
fi
```

```

if [ -z $string1 ]; then
    echo "string1 has a length equal to zero"
else
    echo "string1 has a length greater than zero"
fi

```

If you execute `compare1`, you get the following result:

```

string1 not equal to string2
string2 not equal to string1
string1 is not empty
string2 has a length greater than zero
string1 has a length greater than zero

```

If two strings are not equal in size, the system pads out the shorter string with trailing spaces for comparison. That is, if the value of `string1` is "abc" and that of `string2` is "ab", `string2` is padded with a trailing space for comparison purposes; it has the value "ab " (with a space after the letters).

Numeric Comparison

The following operators can be used to compare two numbers:

- ▶ `-eq`—Compares whether two numbers are equal
- ▶ `-ge`—Compares whether one number is greater than or equal to the other number
- ▶ `-le`—Compares whether one number is less than or equal to the other number
- ▶ `-ne`—Compares whether two numbers are not equal
- ▶ `-gt`—Compares whether one number is greater than the other number
- ▶ `-lt`—Compares whether one number is less than the other number

The following shell program compares three numbers, `number1`, `number2`, and `number3`:

```

#!/bin/sh
number1=5
number2=10
number3=5

if [ $number1 -eq $number3 ]; then
    echo "number1 is equal to number3"
else
    echo "number1 is not equal to number3"
fi

if [ $number1 -ne $number2 ]; then
    echo "number1 is not equal to number2"

```

```

else
    echo "number1 is equal to number2"
fi

if [ $number1 -gt $number2 ]; then
    echo "number1 is greater than number2"
else
    echo "number1 is not greater than number2"
fi

if [ $number1 -ge $number3 ]; then
    echo "number1 is greater than or equal to number3"
else
    echo "number1 is not greater than or equal to number3"
fi

if [ $number1 -lt $number2 ]; then
    echo "number1 is less than number2"
else
    echo "number1 is not less than number2"
fi

if [ $number1 -le $number3 ]; then
    echo "number1 is less than or equal to number3"
else
    echo "number1 is not less than or equal to number3"
fi

```

When you execute the shell program, you get the following results:

```

number1 is equal to number3
number1 is not equal to number2
number1 is not greater than number2
number1 is greater than or equal to number3
number1 is less than number2
number1 is less than or equal to number3

```

File Operators

You can use the following operators as file comparison operators:

- ▶ **-d**—Determines whether a file is a directory
- ▶ **-f**—Determines whether a file is a regular file
- ▶ **-r**—Determines whether read permission is set for a file
- ▶ **-s**—Determines whether a file exists and has a length greater than zero

- ▶ **-w**—Determines whether write permission is set for a file
- ▶ **-x**—Determines whether execute permission is set for a file

Assume that a shell program called `compare3` is in a directory with a file called `file1` and a subdirectory `dir1` under the current directory. Assume that `file1` has a permission of `r-x` (read and execute permission) and `dir1` has a permission of `rwX` (read, write, and execute permission). The code for the shell program would look like this:

```
#!/bin/sh
if [ -d $dir1 ]; then
    echo "dir1 is a directory"
else
    echo "dir1 is not a directory"
fi

if [ -f $dir1 ]; then
    echo "dir1 is a regular file"
else
    echo "dir1 is not a regular file"
fi

if [ -r $file1 ]; then
    echo "file1 has read permission"
else
    echo "file1 does not have read permission"
fi

if [ -w $file1 ]; then
    echo "file1 has write permission"
else
    echo "file1 does not have write permission"
fi

if [ -x $dir1 ]; then
    echo "dir1 has execute permission"
else
    echo "dir1 does not have execute permission"
fi
```

If you execute the shell program, you get the following results:

```
dir1 is a directory
file1 is a regular file
file1 has read permission
file1 does not have write permission
dir1 has execute permission
```

Logical Operators

You use logical operators to compare expressions using Boolean logic—that is, comparing values using characters representing *NOT*, *AND*, and *OR*:

- ▶ `!`—Negates a logical expression
- ▶ `-a`—Logically *ANDS* two logical expressions
- ▶ `-o`—Logically *ORS* two logical expressions

This example named `logic` uses the file and directory mentioned in the previous `compare3` example:

```
#!/bin/sh
if [ -x file1 -a -x dir1 ]; then
    echo file1 and dir1 are executable
else
    echo at least one of file1 or dir1 are not executable
fi

if [ -w file1 -o -w dir1 ]; then
    echo file1 or dir1 are writable
else
    echo neither file1 or dir1 are executable
fi

if [ ! -w file1 ]; then
    echo file1 is not writable
else
    echo file1 is writable
fi
```

If you execute `logic`, it yields the following result:

```
file1 and dir1 are executable
file1 or dir1 are writable
file1 is not writable
```

Comparing Expressions with `tcsh`

As stated earlier, the method for comparing expressions in `tcsh` is different from the method used under `pdksh` and `bash`. The comparison of expression demonstrated in this section uses the syntax necessary for the `tcsh` shell environment.

String Comparison

You can use the following operators to compare two string expressions:

- ▶ `==`—Compares whether two strings are equal
- ▶ `!=`—Compares whether two strings are not equal

The following examples compare two strings, `string1` and `string2`, in the shell program `compare1`:

```
#!/bin/tcsh
set string1 = "abc"
set string2 = "abd"

if (string1 == string2) then
    echo "string1 equal to string2"
else
    echo "string1 not equal to string2"
endif

if (string2 != string1) then
    echo "string2 not equal to string1"
else
    echo "string2 equal to string1"
endif
```

If you execute `compare1`, you get the following results:

```
string1 not equal to string2
string2 not equal to string1
```

Number Comparison

You can use the following operators to compare two numbers:

- ▶ `>=`—Determines whether one number is greater than or equal to the other number
- ▶ `<=`—Determines whether one number is less than or equal to the other number
- ▶ `>`—Determines whether one number is greater than the other number
- ▶ `<`—Determines whether one number is less than the other number

The next examples compare three numbers, `number1`, `number2`, and `number3`, in a shell program called `compare2`:

```
#!/bin/tcsh
set number1=5
set number2=10
set number3=5

if (number1 > number2) then
    echo "number1 is greater than number2"
else
    echo "number1 is not greater than number2"
endif
```

```

if (number1 >= number3) then
    echo "number1 is greater than or equal to number3"
else
    echo "number1 is not greater than or equal to number3"
endif

if (number1 < number2) then
    echo "number1 is less than number2"
else
    echo "number1 is not less than number2"
endif

if (number1 <= number3) then
    echo "number1 is less than or equal to number3"
else
    echo "number1 is not less than or equal to number3"
endif

```

When executing the shell program `compare2`, you get the following results:

```

number1 is not greater than number2
number1 is greater than or equal to number3
number1 is less than number2
number1 is less than or equal to number3

```

File Operators

You can use the following operators as file comparison operators:

- ▶ **-d**—Determines whether a file is a directory
- ▶ **-e**—Determines whether a file exists
- ▶ **-f**—Determines whether a file is a regular file
- ▶ **-o**—Determines whether a user is the owner of a file
- ▶ **-r**—Determines whether read permission is set for a file
- ▶ **-w**—Determines whether write permission is set for a file
- ▶ **-x**—Determines whether execute permission is set for a file
- ▶ **-z**—Determines whether the file size is zero

The following examples are based on a shell program called `compare3`, which is in a directory with a file called `file1` and a subdirectory `dir1` under the current directory. Assume that `file1` has a permission of `r-x` (read and execute permission), and `dir1` has a permission of `rwX` (read, write, and execute permission).

The following is the code for the `compare3` shell program:

```
#!/bin/tcsh
if (-d dir1) then
    echo "dir1 is a directory"
else
    echo "dir1 is not a directory"
endif

if (-f dir1) then
    echo "file1 is a regular file"
else
    echo "file1 is not a regular file"
endif

if (-r file1) then
    echo "file1 has read permission"
else
    echo "file1 does not have read permission"
endif

if (-w file1) then
    echo "file1 has write permission"
else
    echo "file1 does not have write permission"
endif

if (-x dir1) then
    echo "dir1 has execute permission"
else
    echo "dir1 does not have execute permission"
endif

if (-z file1) then
    echo "file1 has zero length"
else
    echo "file1 has greater than zero length"
endif
```

If you execute the file `compare3`, you get the following results:

```
dir1 is a directory
file1 is a regular file
file1 has read permission
file1 does not have write permission
dir1 has execute permission
file1 has greater than zero length
```

Logical Operators

You use logical operators with conditional statements. You use the following operators to negate a logical expression or to perform logical ANDs and ORs:

- ▶ `!`—Negates a logical expression
- ▶ `&&`—Logically ANDs two logical expressions
- ▶ `||`—Logically ORs two logical expressions

This example named `logic` uses the file and directory mentioned in the previous `compare3` example:

```
#!/bin/tcsh
if ( -x file1 && -x dir1 ) then
    echo file1 and dir1 are executable
else
    echo at least one of file1 or dir1 are not executable
endif

if ( -w file1 || -w dir1 ) then
    echo file1 or dir1 are writable
else
    echo neither file1 or dir1 are executable
endif

if ( ! -w file1 ) then
    echo file1 is not writable
else
    echo file1 is writable
endif
```

If you execute `logic`, it yields the following result:

```
file1 and dir1 are executable
file1 or dir1 are writable
file1 is not writable
```

The `for` Statement

You use the `for` statement to execute a set of commands once each time a specified condition is true. The `for` statement has a number of formats. The first format used by `pdksh` and `bash` is as follows:

```
for curvar in list
do
    statements
done
```

You should use this format if you want to execute `statements` once for each value in `list`. For each iteration, the current value of the list is assigned to `curvar`. `list` can be a variable containing a number of items or a list of values separated by spaces. The second format is as follows:

```
for curvar
do
    statements
done
```

In this format, the `statements` are executed once for each of the positional parameters passed to the shell program. For each iteration, the current value of the positional parameter is assigned to the variable `curvar`. You can also write this format as follows:

```
for curvar in $
do
    statements
done
```

Remember that `$@` gives you a list of positional parameters passed to the shell program, quoted in a manner consistent with the way the user originally invoked the command.

Under `tcsh`, the `for` statement is called `foreach`, and the format is as follows:

```
foreach curvar (list)
    statements
end
```

In this format, `statements` are executed once for each value in `list`, and, for each iteration, the current value of `list` is assigned to `curvar`.

Suppose that you want to create a backup version of each file in a directory to a subdirectory called `backup`. You can do the following in `pksh` and `bash`:

```
#!/bin/sh
for filename in *
do
    cp $filename backup/$filename
    if [ $? -ne 0 ]; then
        echo "copy for $filename failed"
    fi
done
```

In this example, a backup copy of each file is created. If the copy fails, a message is generated.

The same example in `tcsh` is as follows:

```
#!/bin/tcsh
foreach filename (`bin/ls`)
```

```

cp $filename backup/$filename
if ($? != 0) then
    echo "copy for $filename failed"
endif
end

```

The while Statement

You can use the `while` statement to execute a series of commands while a specified condition is `true`. The loop terminates as soon as the specified condition evaluates to `false`. It is possible that the loop will not execute at all if the specified condition initially evaluates to `false`. You should be careful with the `while` command because the loop never terminates if the specified condition never evaluates to `false`.

ENDLESS LOOPS HAVE A PLACE IN SHELL PROGRAMS

Endless loops can sometimes be useful. For example, you can easily construct a simple command that constantly monitors the 802.11 link quality of a network interface by using a few lines of script:

```

#!/bin/sh
while :
do
    /sbin/iwconfig wlan0 | grep Link | tr '\n' '\r'
Done

```

The script outputs the search, and then the `tr` command formats the output. The result is a simple animation of a constantly updated single line of information:

```
Link Quality:92/92 Signal level:-11 dBm Noise level:-102 dBm
```

You can also use this technique to create a graphical monitoring client for X that outputs traffic information and activity about a network interface:

```

#!/bin/sh
xterm -geometry 75x2 -e \
bash -c \
"while ;; do \
    /sbin/ifconfig eth0 | \
    grep 'TX bytes' | \
    tr '\n' '\r' ; \
done"

```

This simple example uses a `bash` command-line script (enabled by `-c`) to execute a command line repeatedly. The command line pipes the output of the `ifconfig` command through `grep`, which searches the output of `ifconfig` and then pipes a line containing the string "TX bytes" to the `tr` command. The `tr` command then removes the carriage return at the end of the line to display the information inside an `/xterm` X11 terminal window, automatically sized by the `-geometry` option:

```
RX bytes:4117594780 (3926.8 Mb) TX bytes:452230967 (431.2 Mb)
```

Endless loops can be so useful that Linux includes a command that repeatedly executes a given command line. For example, you can get a quick report about a system's hardware health by using the `sensors` command. Instead of using a shell script to loop the output endlessly, you can use the `watch` command to repeat the information and provide simple animation:

```
matthew@seymour:~$ watch "sensors -f | cut -c 1-20"
```

In `pdksh` and `bash`, use the following format for the `while` flow control construct:

```
while expression
do
    statements
done
```

In `tcsh`, use the following format:

```
while (expression)
    statements
End
```

If you want to add the first five even numbers, you can use the following shell program in `pdksh` and `bash`:

```
#!/bin/bash
loopcount=0
result=0
while [ $loopcount -lt 5 ]
do
    loopcount=`expr $loopcount + 1`
    increment=`expr $loopcount \* 2`
    result=`expr $result + $increment`
doneecho "result is $result"
```

In `tcsh`, you can write this program as follows:

```
#!/bin/tcsh
set loopcount = 0
set result = 0
while ($loopcount < 5)
    set loopcount = `expr $loopcount + 1`
    set increment = `expr $loopcount \* 2`
    set result = `expr $result + $increment`

end

echo "result is $result"
```

The `until` Statement

You can use the `until` statement to execute a series of commands until a specified condition is `true`.

The loop terminates as soon as the specified condition evaluates to `true`.

In `pdksh` and `bash`, the following format is used:

```
until expression
do
    statements
done
```

As you can see, the format of the `until` statement is similar to that of the `while` statement, but the logic is different: In a `while` loop, you execute until an expression is `false`, whereas in an `until` loop, you loop until the expression is `true`. An important part of this difference is that `while` is executed zero or more times (so it is potentially not executed at all), but `until` is repeated one or more times, meaning it is executed at least once.

If you want to add the first five even numbers, you can use the following shell program in `pdksh` and `bash`:

```
#!/bin/bash
loopcount=0
result=0
until [ $loopcount -ge 5 ]
do
    loopcount=`expr $loopcount + 1`
    increment=`expr $loopcount \* 2`
    result=`expr $result + $increment`
done

echo "result is $result"
```

The example here is identical to the example for the `while` statement except that the condition being tested is just the opposite of the condition specified in the `while` statement.

The `tcsh` shell does not support the `until` statement.

The `repeat` Statement (`tcsh`)

You use the `repeat` statement to execute only one command a fixed number of times.

If you want to print a hyphen (-) 80 times with one hyphen per line on the screen, you can use the following command:

```
repeat 80 echo '-'
```

The `select` Statement (pdksh)

You use the `select` statement to generate a menu list if you are writing a shell program that expects input from the user online. The format of the `select` statement is as follows:

```
select item in itemlist
do
    Statements
done
```

`itemlist` is optional. If it isn't provided, the system iterates through the `item` entries one at a time. If `itemlist` is provided, however, the system iterates for each entry in `itemlist`, and the current value of `itemlist` is assigned to `item` for each iteration, which then can be used as part of the statements being executed.

If you want to write a menu that gives the user a choice of picking a `Continue` or a `Finish`, you can write the following shell program:

```
#!/bin/ksh
select item in Continue Finish
do
    if [ $item = "Finish" ]; then
        break
    fi
done
```

When the `select` command is executed, the system displays a menu with numeric choices—in this case, 1 for `Continue` and 2 for `Finish`. If the user chooses 1, the variable `item` contains the value `Continue`; if the user chooses 2, the variable `item` contains the value `Finish`. When the user chooses 2, the `if` statement is executed, and the loop terminates.

The `shift` Statement

You use the `shift` statement to process the positional parameters, one at a time, from left to right. Recall that the positional parameters are identified as `$1`, `$2`, `$3`, and so on. The effect of the `shift` command is that each positional parameter is moved one position to the left, and the current `$1` parameter is lost.

The `shift` statement is useful when you are writing shell programs in which a user can pass various options. Depending on the specified option, the parameters that follow can mean different things or might not be there at all.

The format of the `shift` command is as follows:

```
shift number
```

The parameter `number` is the number of places to be shifted and is optional. If it is not specified, the default is 1; that is, the parameters are shifted one position to the left. If it is specified, the parameters are shifted `number` positions to the left.

The `if` Statement

The `if` statement evaluates a logical expression to make a decision. An `if` condition has the following format in `pdksh` and `bash`:

```
if [ expression ]; then
    Statements
elif [ expression ]; then
    Statements
else
    Statements
fi
```

`if` conditions can be nested. That is, an `if` condition can contain another `if` condition within it. It isn't necessary for an `if` condition to have an `elif` or `else` part. The `else` part is executed if none of the expressions that are specified in the `if` statement are true and are not evaluated if preceding `elif` statements are true. The word `fi` is used to indicate the end of the `if` statements, which is very useful if you have nested `if` conditions. In such a case, you should be able to match `fi` to `if` to ensure that all `if` statements are properly coded.

In the following example for `bash` or `pdksh`, a variable `var` can have either of two values: `Yes` or `No`. Any other value is invalid. This can be coded as follows:

```
if [ $var = "Yes" ]; then
    echo "Value is Yes"
elif [ $var = "No" ]; then
    echo "Value is No"
else
    echo "Invalid value"
fi
```

In `tcsh`, the `if` statement has two forms. The first form, similar to the one for `pdksh` and `bash`, is as follows:

```
if (expression) then
    Statements
else if (expression) then
    Statements
else
    Statements
endif
```

Using the example of the variable `var` having only two values, `Yes` and `No`, here is how it is coded with `tcsh`:

```
if ($var == "Yes") then
    echo "Value is Yes"
```



```

else if ($var == "No" ) then
    echo "Value is No"
else
    echo "Invalid value"
endif

```

The second form of the `if` condition for `tcsh` is as follows:

```
if (expression) command
```

In this format, only a single command can be executed if the expression evaluates to `true`.

The `expr` Statement

You use `expr` to evaluate an expression. It can evaluate numbers and mathematic expressions. It can also be used to evaluate strings and perform comparisons. Some examples follow.

To add or subtract, multiply or divide, use one of the following:

```

expr 12 + 2
expr 26-24
expr 6 \* 3
expr 11 / 3

```

There are two things to note in this set of operations.

- ▶ When you multiply, you must use an escape character before the multiplication operator for the command to parse correctly, which is why you see `\` (the escape character) before the `*` (the multiplication character).
- ▶ When you divide, only the whole number result will be returned, so, for example, `expr 11 / 2` returns 3. To get the remainder, use the following, which returns 2:

```
expr 11 / 2
```

A number of string-related operations are available. Each involves adding a second word to the command before the string to be evaluated.

To find the length of a string, use the following:

```
expr length string
```

If the string includes spaces, you must again use the escape character, like this:

```

expr length linux\ is\ cool
13

```

You can do many more things with `expr`. Read the `man` page for more information.

The case Statement

You use the `case` statement to execute statements depending on a discrete value or a range of values matching the specified variable. In most cases, you can use a `case` statement instead of an `if` statement if you have a large number of conditions.

The format of a `case` statement for `pksh` and `bash` is as follows:

```
case str in
  str1 | str2)
    Statements;;
  str3|str4)
    Statements;;
  *)
    Statements;;
esac
```

You can specify a number of discrete values—such as `str1`, `str2`, and so on—for each condition, or you can specify a value with a wildcard. The last condition should be an asterisk (*) and is executed if none of the other conditions are met. For each of the specified conditions, all the associated statements until the double semicolon (;;) are executed.

You can write a script that echoes the name of the month if you provide the month number as a parameter. If you provide a number that isn't between 1 and 12, you get an error message. The script is as follows:

```
#!/bin/sh

case $1 in
  01 | 1) echo "Month is January";;
  02 | 2) echo "Month is February";;
  03 | 3) echo "Month is March";;
  04 | 4) echo "Month is April";;
  05 | 5) echo "Month is May";;
  06 | 6) echo "Month is June";;
  07 | 7) echo "Month is July";;
  08 | 8) echo "Month is August";;
  09 | 9) echo "Month is September";;
  10) echo "Month is October";;
  11) echo "Month is November";;
  12) echo "Month is December";;
  *) echo "Invalid parameter";;
esac
```

You need to end the statements under each condition with a double semicolon (;;). If you do not, the statements under the next condition are also executed.

The format for a `case` statement for `tcsh` is as follows:

```
switch (str)
  case str1|str2:
    Statements
    breaksw
  case str3|str4:
    Statements
    breaksw
  default:
    Statements
    breaksw
endsw
```

You can specify a number of discrete values—such as `str1`, `str2`, and so on—for each condition, or you can specify a value with a wildcard. The last condition should be the default and is executed if none of the other conditions are met. For each of the specified conditions, all the associated statements until `breaksw` are executed.

You can write the example that echoes the month when a number is given, shown earlier for `pdksh` and `bash`, in `tcsh` as follows:

```
#!/bin/tcsh

set month = 5
switch ( $month )
  case 1: echo "Month is January" ; breaksw
  case 2: echo "Month is February" ; breaksw
  case 3: echo "Month is March" ; breaksw
  case 4: echo "Month is April" ; breaksw
  case 5: echo "Month is May" ; breaksw
  case 6: echo "Month is June" ; breaksw
  case 7: echo "Month is July" ; breaksw
  case 8: echo "Month is August" ; breaksw
  case 9: echo "Month is September" ; breaksw
  case 10: echo "Month is October" ; breaksw
  case 11: echo "Month is November" ; breaksw
  case 12: echo "Month is December" ; breaksw
  default: echo "Oops! Month is October!" ; breaksw
endsw
```

You need to end the statements under each condition with `breaksw`. If you do not, the statements under the next condition are also executed.

The `break` and `exit` Statements

You should be aware of two other statements: the `break` statement and the `exit` statement.

You can use the `break` statement to terminate an iteration loop, such as a `for`, `until`, or `repeat` command.

You can use `exit` statement to exit a shell program. You can optionally use a number after `exit`. If the current shell program has been called by another shell program, the calling program can check for the code (the `$?` or `$status` variable, depending on shell) and make a decision accordingly.

Using Functions in Shell Scripts

As with other programming languages, in Ubuntu shell programs also support functions. A function is a piece of a shell program that performs a particular process; you can reuse the same function multiple times within the shell program. Functions help eliminate the need for duplicating code as you write shell programs.

The following is the format of a function in `pdksh` and `bash`:

```
func() {
    Statements
}
```

You can call a function as follows:

```
func param1 param2 param3
```

The parameters `param1`, `param2`, and so on are optional. You can also pass the parameters as a single string—for example, `$@` . A function can parse the parameters as if they were positional parameters passed to a shell program from the command line as command-line arguments but instead use values passed inside the script. For example, the following script uses a function named `Displaymonth()` that displays the name of the month or an error message if you pass a month number out of the range 1 to 12. This example works with `pdksh` and `bash`:

```
#!/bin/sh
Displaymonth() {
    case $1 in
        01 | 1) echo "Month is January";;
        02 | 2) echo "Month is February";;
        03 | 3) echo "Month is March";;
        04 | 4) echo "Month is April";;
        05 | 5) echo "Month is May";;
        06 | 6) echo "Month is June";;
        07 | 7) echo "Month is July";;
```

```
08 | 8) echo "Month is August";;
09 | 9) echo "Month is September";;
10) echo "Month is October";;
11) echo "Month is November";;
12) echo "Month is December";;
*) echo "Invalid parameter";;
esac
}
Displaymonth 8
```

The preceding program displays the following output:

```
Month is August
```

References

- ▶ www.gnu.org/software/bash/—The `bash` home page at the GNU Software Project
- ▶ www.tldp.org/LDP/abs/html/—Mendel Cooper’s “Advanced Bash-Scripting Guide”
- ▶ www.freeos.com/guides/lsst/—Linux shell scripting tutorial
- ▶ <https://kornshell.com>—The KornShell website
- ▶ <https://web.cs.mun.ca/~michael/pdksh/>—The `pdksh` home page
- ▶ www.tcsh.org—Find out more about `tcsh` here
- ▶ www.zsh.org/—Examine `zsh` in more detail here

CHAPTER 15

The Boot Process

In this chapter, you learn about making tasks into services that run as your system starts and making them into services you can start and stop by hand. You also learn about the entire boot process.

After you turn on the power switch, the boot process begins with the computer executing code stored in a chip called the *BIOS*, or *basic input/output system*; this process occurs no matter what operating system you have installed. The Linux boot process begins when the code known as the *boot loader* starts loading the Linux kernel and ends when the login prompt appears.

As a system administrator, you will use the skills you learn in this chapter to control your system's services and manage runlevels on your computer. Understanding the management of the system services and states is essential to understanding how Linux works (especially in a multiuser environment) and helps untangle the mysteries of a few of your Ubuntu system's configuration files. Furthermore, a good knowledge of the `crond` daemon that handles task scheduling is essential for administrators at all skill levels, so you will want to combine this knowledge with what you learned in Chapter 14, "Automating Tasks and Shell Scripting."

Running Services at Boot

Although most people consider a computer to be either on or off, in Ubuntu and Linux in general, there are a number of states in between. Known as *runlevels*, they define what system services are started upon boot. These services are simply applications running in the background that provide some needed function to a system, such as getting information from the mouse and sending it to the display;

IN THIS CHAPTER

- ▶ Running Services at Boot
- ▶ Beginning the Boot Loading Process
- ▶ Loading the Linux Kernel
- ▶ Starting and Stopping Services with `systemd`
- ▶ Boot-Repair
- ▶ References

or a service could monitor the partitions to see whether they have enough free space left on them. Services are typically loaded and run (also referred to as being started) during the boot process, in the same way as Microsoft Windows services are. For a while, Ubuntu used a system known as Upstart instead of the classic and venerable SysVinit; Upstart had a special backward-compatibility layer that could use runlevels in the way that Linux veterans are accustomed to doing for services not otherwise handled by Upstart. Starting in 2015, Ubuntu switched to another system called `systemd`. You'll learn more about runlevels, Upstart, and `systemd` later in this chapter.

Init SYSTEMS

There are many ways to describe `init` systems, but this quote from a thread at Reddit.com is hard to beat (capitalization and spelling errors are from the original):

“To understand the fuss, there are 3 `init` systems that you should be aware of: `sysvinit`, `upstart` and `systemd`. `SysVinit` is really old, outdated and only allows for a sequential startup of services, that is, all services started by `init` must wait for the previous service to have completed their startup process before the next can startup. Both `upstart` and `systemd` are designed to tackle the limitations of `init`, and allows for concurrent service initialization, that is, multiple services can startup at the same time, as long as they are not dependent on each other, and allows taking advantage of multi core processing. They also allow for services to startup and shutdown for specific events, such as a network connection going up or another service has started. `Upstart` was developed by Canonical was initially released in 2006 and was used in Ubuntu, Debian, Fedora, RHEL, CentOS, and many others. `Systemd` is a much younger system that was initially released in 2011, however, most major distros has already migrated to using `systemd` by default.

“Both `Upstart` and `Systemd` are event based, however their architecture and configuration setup are very different and this is where the worrying comes from. If the configurations aren't ported correctly then systems are going to behave differently to what's expected and this can lead to issues for many users.”

—hitsujitMO in www.reddit.com/r/Ubuntu/comments/2yeyyi/grab_your_pitchforks_ubuntu_to_switch_to_systemd/cp92iro

You can manage nearly every aspect of your computer and how it behaves after booting via configuring and ordering boot scripts and by using various system administration utilities included with Ubuntu. In this chapter, you learn how to work with these boot scripts and system administration utilities. This chapter also offers advice for troubleshooting and fixing problems that might arise with software configuration or the introduction or removal of various types of hardware from your system.

Beginning the Boot Loading Process

Although the actual boot loading mechanism for Linux varies on different hardware platforms (such as SPARC, Alpha, and PowerPC systems), Intel-based PCs running Ubuntu most often use the same mechanism throughout product lines. This process is traditionally accomplished through a BIOS. The BIOS is an application stored in a chip on the motherboard that initializes the hardware on the motherboard (and often the hardware

that's attached to the motherboard). The BIOS gets the system ready to load and run the software that we recognize as the operating system.

As a last step, the BIOS code looks for a special program known as the boot loader or boot code. The instructions in this little bit of code tell the BIOS where the Linux kernel is located, how it should be loaded into memory, and how it should be started.

If all goes well, the BIOS looks for a bootable volume such as a CD-ROM, hard drive, RAM disk, USB drive, or other media. The bootable volume contains a special hexadecimal value written to the volume by the boot loader application (such as Ubuntu's default, GRUB2) when the boot loader code was first installed in the system's drives. The BIOS searches volumes in the order established by the BIOS settings (for example, USB first, followed by a DVD-ROM, and then a hard drive) and then boots from the first bootable volume it finds. Modern BIOSs allow considerable flexibility in choosing the device used for booting the system.

NOTE

If the BIOS detects a hardware problem, the boot process fails, and the BIOS generates a few beeps from the system speaker. These "beep codes" indicate the nature of the problem the BIOS has encountered. The codes vary among manufacturers, and the diagnosis of problems occurring during this phase of the boot process is beyond the scope of this book and does not involve Linux. If you encounter a problem, consult the motherboard manual or contact the manufacturer of the motherboard. Another good source for learning about beep codes is www.computerhope.com/beep.htm.

Next, the BIOS looks on the bootable volume for boot code in the partition boot sector, also known as the *master boot record (MBR)*, of the first hard disk. The MBR contains the boot loader code and the partition table; you can think of it as an index for a book, plus a few comments on how to start reading the book. If the BIOS finds a boot loader, it loads the boot loader code into memory. At that point, the BIOS's job is completed, and it passes control of the system to the boot loader.

As computing evolved, the BIOS began to be a limiting factor because some of its limitations were not easy to overcome. Intel was the first company to notice this as it developed its Itanium systems in the late 1990s. That work eventually became the foundation for the Unified Extensible Firmware Interface (UEFI).

UEFI serves a similar role to BIOS and has replaced BIOS in most modern systems. For most end users, the difference is negligible, except that if you have a UEFI computer, you may need to do some research to install Ubuntu. UEFI firmware often has a BIOS mode that allows the firmware to work like a traditional BIOS and that is more easily compatible, especially if you are installing Ubuntu as the sole operating system on the computer rather than as a dual-boot. There were some significant problems early on, but they seem to have lessened as the Linux community has learned more about UEFI and made things more easily compatible. To learn more, especially if you are having trouble, see <https://help.ubuntu.com/community/UEFI>.

One interesting aspect of using UEFI involves hard disk partitioning. When UEFI is used with Windows 8.x or 10, often the partitioning is set up using GPT (GUID Partition Table) rather than the existing MBR standard. You can use GPT with Ubuntu, but if only Ubuntu is to be installed on the drive, reformatting the hard disk to use MBR is often easier. As with BIOS and UEFI, most people do not need to know or care whether they are using MBR or GPT. To learn more, see www.howtogeek.com/193669/whats-the-difference-between-gpt-and-mbr-when-partitioning-a-drive/. If the system has a UEFI BIOS, GRUB is stored as an EFI application on the EFI system partition and not the MBR/GPT.

The boot loader locates the Linux kernel on the disk and loads it into memory. After that task is completed, the boot loader passes control of the system to the Linux kernel. You can see how one process builds on another in an approach that enables many different operating systems to work with the same hardware.

NOTE

Linux is very flexible and can be booted from multiple images on a CD-ROM, over a network using PXE (pronounced “pixie”) or NetBoot, or on a headless server with the console display sent over a serial or network connection. Work is even under way at www.coreboot.org to create a special Linux BIOS that will expedite the boot process because Linux does not need many of the services offered by the typical BIOS.

This kind of flexibility enables Linux to be used in a variety of ways, such as for remote servers or diskless workstations, which are not generally seen in personal home use.

Loading the Linux Kernel

In a general sense, the kernel manages the system resources. As the user, you do not often interact with the kernel, but instead you interact with the applications you are using. Linux refers to each application as a *process*, and the kernel assigns each process a number called a *process ID (PID)*. Traditionally, the Linux kernel loads and runs a process named `init`, which is also known as the “ancestor of all processes” because it starts every subsequent process. The traditional `init` system was SysVinit. It has been replaced by newer options. One of these options was Upstart, which was written by Ubuntu developers and made available for any distribution to use. Upstart was replaced by `systemd` as of Ubuntu 15.04. This chapter walks through the traditional SysVinit method first and then provides details on Upstart and `systemd`.

The next step of the boot process traditionally begins with a message that the Linux kernel is loading and a series of messages that are printed to the screen, giving the status of each command. A failure should display an error message. The `quiet` option may be passed to the kernel at boot time to suppress many of these messages. Ubuntu does not display these messages by default but instead uses a boot process created by the Fedora/Red Hat developers called Plymouth that is fast and incorporates a beautiful boot screen.

If the boot process were halted at this point, the system would just sit idle, and the screen would be blank. To make the system useful for users, you need to start the system services. Those services are some of the applications that enable you to interact with the system.

Starting and Stopping Services with `systemd`

Ubuntu uses `systemd` as a modern replacement for `init`. It was created by Red Hat and has seen near-universal adoption across Linux distributions. The job of `systemd` is to manage services that run on the system and is comprised of a set of basic building blocks.

`systemd` is the mother of all processes and is responsible for bringing the Linux kernel up to a state where work can be done. To do this, it starts by mounting the file systems that are defined in `/etc/fstab`, including swap files or partitions.

From here, the configuration files in `/etc` are available and are loaded, starting with the configuration file for `systemd` itself at `/etc/systemd/system/default.target`, which is actually just a symbolic link to the actual `.target` file, which depends on how the system is set up.

For a typical desktop system, that symlink will point to the `graphical.target` file. For a server, it will likely point to `multi-user.target`, which is command line only. There are other target files, which loosely correspond to SystemV runlevels, as shown in Table 15.1.

Table 15.1 `Systemd` Targets and How They Relate to SystemV Runlevels

<code>systemd</code> Target	<code>systemd</code> Target Alias	SystemV Runlevel	Description
<code>default.target</code>			This is a symlink to whichever of the following targets is set to load by default.
<code>graphical.target</code>	<code>runlevel5.target</code>	5	For a desktop system, this is the default and loads a multiuser system with a GUI.
<code>multi-user.target</code>	<code>runlevel3.target</code>	3	This will load and run all services for a multiuser system but is command line only. This is typical for a server.
<code>emergency.target</code>		S	Loads single-user mode with no running services, no mounted file systems, and only an emergency shell.
<code>rescue.target</code>	<code>runlevel1.target</code>	1	Loads a base system with basic services running, mounted file systems, and a rescue shell.
<code>halt.target</code>			Halts the system without powering down.
<code>poweroff.target</code>	<code>runlevel0.target</code>	0	Halts the system and powers down.
<code>reboot.target</code>	<code>runlevel6.target</code>	6	Reboots the system.
	<code>runlevel2.target</code>	2	Boots a multiuser system with all non-GUI services running except NFS (network file system).
	<code>runlevel4.target</code>	4	Unused by <code>systemd</code> .

Using these targets and target aliases, you can change the running mode similar to how in the past we would change runlevels using SystemV init commands. When `systemd` is engaged during the boot process, it uses the set target to discover and load expected dependencies, the services required to run the Linux system at the expected level of functionality.

To change the system state using target files, use the following commands (see Table 15.2):

Table 15.2 `systemd` State-Related Commands

Command	Description
<code>systemctl get-default</code>	Lists the currently set default target
<code>sudo systemctl set-default new.target</code>	Changes the default to a different target; for example, replace <code>new</code> with something from the preceding list, but never set the default to <code>halt.target</code> , <code>poweroff.target</code> , or <code>reboot.target</code>
<code>sudo systemctl isolate new.target</code>	Changes to a different target; this is like changing runlevel in SystemV.
<code>sudo systemctl list-dependencies some.target</code>	Lists the dependencies of a specific target, which is a good idea before you run an <code>isolate</code> command to make sure you will have what you require after you switch.

After the base services are loaded and running, `systemd` will also load any user-defined services. All of these services are called units. To list every available unit file on your system, use

```
$ systemctl list-unit-files
```

A word of warning: The output will include hundreds of unit files in the listing. On the positive side, it will tell you whether each one is enabled, disabled, or in some other state. States include

- ▶ **disabled**, which means it is not currently available.
- ▶ **enabled**, which means it is currently available.
- ▶ **masked**, which means the unit file has been marked as unstartable and is linked to `/dev/null`. This prevents a unit file from being started, either automatically or manually, until it is unmasked.
- ▶ **static**, which means the unit file does not include an “install” section, which is necessary to enable a unit; these are usually not meant to be run by users and generally perform a one-time action in the startup process or work as a dependency of another unit and are not intended to be run alone. To interact with `systemd`, you typically use the `systemctl` command. The main interactions you are likely to have with `systemd` involve starting and stopping services. Table 15.3 shows commands related to services.

Table 15.3 `systemd` Service-Related Commands

Command	Description
<code>systemctl</code>	List running services
<code>systemctl start</code> <code>servicename service</code>	Start a service
<code>systemctl stop</code> <code>servicename</code> <code>service</code>	Stop a service
<code>systemctl restart</code> <code>servicename service</code>	Restart a service
<code>systemctl reload</code> <code>servicename service</code>	Reload a service (this only tells the service to reload its configuration files, rather than restart the entire service; typically, restarting is a safer option)
<code>systemctl status</code> <code>servicename service</code>	Show the status of a service
<code>systemctl condrestart</code> <code>servicename service</code>	Restart a service if it is already running
<code>systemctl enable</code> <code>servicename service</code>	Enable a service at startup
<code>systemctl disable</code> <code>servicename service</code>	Disable a service at startup (typically to remove a service from the list of those starting automatically at boot)
<code>systemctl mask</code> <code>servicename</code> <code>service</code>	Make a service unavailable for use
<code>systemctl unmask</code> <code>servicename service</code>	Make a service available for use again

A few standard commands are replaced in `systemd`, including the ones listed in Table 15.4.

Table 15.4 Other `systemd` Commands

Command	Description
<code>systemctl halt</code>	Halt the system
<code>systemctl reboot</code>	Reboot the system
<code>systemctl cat</code> <code>servicename</code>	Display the unit file for a service
<code>systemctl list-dependencies</code> <code>servicename</code>	Display the dependencies for a service
<code>systemctl show</code> <code>servicename</code>	Display the low-level properties of a service, anything that is set using a key=value format
<code>sudo systemctl edit</code> <code>servicename</code>	Temporarily edit a service file just in memory
<code>sudo systemctl edit --full</code> <code>servicename</code>	Permanently edit a service file; to remove any changes you have made, delete the unit's <code>.d</code> configuration directory or the modified service file from <code>/etc/systemd/system</code> and the default file will be used
<code>journalctl -f</code>	Follow the system log file; replaces <code>tail -f /var/log/message</code>

Services are defined in `systemd` unit files, which end with `.service`. Many examples of these are found in `/lib/systemd/system`.

There is much more to learn about `systemd` than is appropriate for a book with such a wide range of topics to cover—especially as few people need to know more than is covered here. See the official documentation listed in “References” section to learn more if you find yourself in need.

Controlling Services at Boot with Administrative Tools

You can configure what services run at startup with Startup Applications Preferences (shown in Figure 15.1). Here Ubuntu lists all the services that you can have automatically start at boot time. Checked programs are enabled by default, but you can uncheck the ones you don’t want. Disabling services randomly “to make things go faster” is not recommended. Some services might be vital for the continuing operation of your computer, such as the graphical login manager and the system communication bus.

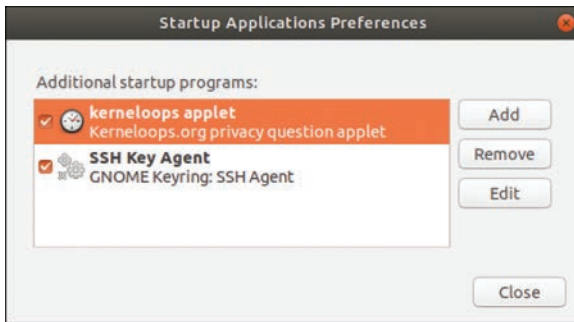


FIGURE 15.1 You can enable and disable Ubuntu’s boot services by toggling the check boxes in the Startup Applications Preferences dialog box.

Troubleshooting Runlevel Problems

Reordering or changing system services is rarely necessary when using Ubuntu unless some disaster occurs, but system administrators should have a basic understanding of how Linux boots and how services are controlled to perform troubleshooting or diagnose problems. By using utilities such as the `dmesg | less` command to read kernel output after booting or by examining system logging with `cat /var/log/syslog | less`, it is possible to gain a bit more detail about what is going on when faced with troublesome drivers or service failure.

Boot-Repair

Sometimes, such as when you install both Windows and Ubuntu on the same hard drive, boot problems can develop. Boot-Repair is a simple GUI tool you can use to fix such problems. Typically, it just reinstalls GRUB2, but using Boot-Repair is a much easier solution for many users. The program is not yet in the Ubuntu repositories, although plans

for including it have been discussed. In the meanwhile, should you need it, take a look at <https://help.ubuntu.com/community/Boot-Repair> or the official documentation at <https://sourceforge.net/projects/boot-repair/>.

References

- ▶ **`/usr/src/linux/init/main.c`**—A file that appears on your computer after you install the kernel source code, which is the best place to learn about how Linux boots
- ▶ **<https://help.ubuntu.com/community/Grub2>**—Ubuntu community documentation for GRUB2
- ▶ **<https://wiki.freedesktop.org/www/Software/systemd/>**—The official system documentation

This page intentionally left blank

CHAPTER 16

System-Monitoring Tools

To keep your system in optimum shape, you need to be able to monitor it closely. This is imperative in a corporate environment, where uptime is vital and any system failures and downtime can be quite expensive. Whether for checking processes for errant daemons or keeping a close eye on CPU and memory usage, Ubuntu provides a wealth of utilities designed to give you as little or as much feedback as you want. This chapter looks at some of the basic monitoring tools, along with some tactics designed to keep your system up longer. Some of the monitoring tools cover network connectivity, memory, and hard drive usage, and in this chapter you learn how to manipulate active system processes using a mixture of graphical and command-line tools.

Console-Based Monitoring

Those familiar with UNIX system administration already know about the `ps`, or process display, command commonly found on most flavors of UNIX. Because of the close relationship between Linux and UNIX, Ubuntu also includes this command, which enables you to see the current processes running on the system, who owns them, and how resource-hungry they are.

Although the Linux kernel has its own distinct architecture and memory management, it also benefits from enhanced use of the `/proc` file system, the virtual file system found on many UNIX flavors. Through the `/proc` file system, you can communicate directly with the kernel to get a deep view of what is currently happening. Developers tend to use the `/proc` file system as a way of extracting information from the kernel and for their programs to manipulate that information into human-readable formats. A full discussion of the `/proc` file system is beyond the scope of this book. To get a better idea of what it contains, you can take a look at

IN THIS CHAPTER

- ▶ Console-Based Monitoring
- ▶ Graphical Process- and System-Management Tools
- ▶ KDE Process- and System-Monitoring Tools
- ▶ Enterprise Server Monitoring
- ▶ References

<https://en.tldp.org/LDP/Linux-Filesystem-Hierarchy/html/proc.html>, which provides an excellent and in-depth guide.

Processes can also be controlled at the command line, which is important because you might sometimes have only a command-line interface. Whenever an application or a command is launched, either from the command line or a clicked icon, the process that comes from the kernel is assigned an identification number called a *process ID (PID)*. This number is shown in the shell if the program is launched via the command line:

```
matthew@seymour:~$ gedit &
[1] 9649
```

In this example, `gedit` has been launched in the background, and the (`bash`) shell reported a shell job number (`[1]` in this case). A job number or job control is a shell-specific feature that allows a different form of process control, such as sending or suspending programs to the background and retrieving background jobs to the foreground. (See your shell's man pages for more information if you are not using `bash`.)

The second number displayed (`9649` in this example) represents the PID. You can get a quick list of your processes by using the `ps` command, like this:

```
matthew@seymour:~$ ps
  PID TTY          TIME CMD
 9595 pts/0        00:00:00 bash
 9656 pts/0        00:00:00 gedit
 9657 pts/0        00:00:00 ps
```

As you can see, the output includes the PID along with other information, such as the name of the running program. As with any other UNIX command, many options are available; the `proc` man page has a full list. One useful option is `-e`, which lists all processes running on the system. Another is `aux`, which provides a more detailed list of all the processes. You should also know that `ps` works not by polling memory but through the interrogation of the Linux `/proc`, or process file system.

The `/proc` directory contains many files, some of which include constantly updated hardware information (such as battery power levels). Linux administrators often pipe the output of `ps` through `grep` to display information about a specific program, like this:

```
matthew@seymour:~$ ps aux | grep bash
matthew      9656  0.0  0.1 21660  4460 pts/0    Ss   11:39   0:00 bash
```

This example returns the owner (the user who launched the program) and the PID, along with other information such as the percentage of CPU and memory usage, the size of the command (code, data, and stack), the time (or date) the command was launched, and the

name of the command for any process that includes the match `bash`. Processes can also be queried by PID as follows:

```
matthew@seymour:~$ ps 9656
  PID TTY          STAT TIME  COMMAND
 9656 pts/0    S      0:00  gedit
```

You can use the PID to stop a running process by using the shell's built-in `kill` command. This command asks the kernel to stop a running process and reclaim system memory. For example, to stop `gedit` in the preceding example, use the `kill` command like this:

```
matthew@seymour:~$ kill 9656
```

After you press Enter and then press Enter again, the shell reports the following:

```
[1]+  Terminated          gedit
```

Note that users can kill only their own processes, but root can kill all users' processes. Controlling any other running process requires root permission, which you should use judiciously (especially when forcing a kill by using the `-9` option); by inadvertently killing the wrong process through a typo in the command, you could bring down an active system.

Using the `kill` Command to Control Processes

The `kill` command is a basic UNIX system command. You can communicate with a running process by entering a command into its interface, such as when you type into a text editor. But some processes (usually system processes rather than application processes) run without such an interface, and you need a way to communicate with them as well, so you use a system of signals. The `kill` system accomplishes that by sending a signal to a process, and you can use it to communicate with any process. The general format of the `kill` command is as follows:

```
matthew@seymour:~$ kill option PID
```

Note that if you are using `kill` on a process you do not own, you need to have super user privileges and preface the `kill` command with `sudo`.

A number of signal options can be sent as words or numbers, but most are of interest only to programmers. One of the most common is the one used previously to kill `gedit`:

```
matthew@seymour:~$ kill PID
```

This tells the process with `PID` to stop (where you supply the actual PID). Issuing the command without a signal option issues the default, which is `kill -15` (more on that later), and gives us no guarantee that a process will be killed because programs can catch, block, or ignore some terminate signals (and this is a good thing, done by design).

The following example includes a signal for `kill` that cannot be caught (9 is the number of the `SIGKILL` signal):

```
matthew@seymour:~$ kill -9 PID
```

You can use this combination when the plain `kill` shown previously does not work. Be careful, though. Using this does not allow a process to shut down gracefully, and shutting down gracefully is usually preferred because it closes things that the process might have been using and ensures that things such as logs are written before the process disappears. Instead, try this first:

```
matthew@seymour:~$ kill -1 PID
```

This is the signal to “hang up”—stop—and then clean up all associated processes as well (1 is the number of the `SIGHUP` signal).

In fact, some system administrators and programmers prefer something like this progression of signals:

- ▶ `kill -15`—This command sends a `SIGTERM`, which is a clean shutdown that flushes data that needs to be written to disk, cleans up memory registers, and closes the `PID`.
- ▶ `kill -1`—As mentioned earlier, this command sends a `SIGHUP`, which cleans up and usually also causes the program to restart.
- ▶ `kill -2`—This command sends a `SIGINT`, which is an interrupt from the keyboard, the equivalent to sending `Ctrl+C`. For example, if you want to stop a program that is running in the background as a daemon instead of in the terminal foreground, this is a good way to do it.
- ▶ `kill -11`—This command sends a `SIGSEGV`, which causes the problem to experience a segmentation fault and close. It does not flush data to disk, but it may create a core dump file that could be useful for debugging and learning why the program is misbehaving (or behaving exactly as you told it to behave and not as you intended it to behave).
- ▶ `kill -9`—This command sends a `SIGKILL`, which should be used as a last resort because it does not sync any data. Nothing is written to disk—no logging, no debugging, nothing. You stop the `PID` (usually, but not always), but you get nothing that helps you either save data that needed to be written to disk or assists you in figuring out what happened.

As you become proficient at process control and job control, you will learn the utility of a number of `kill` options. You can find a full list of signal options in the `kill` man page.

USING `killall`

The `killall` command allows you to kill a process by name, as in `killall gedit`, which would kill any and all `gedit` processes that are currently running. You can also kill all processes being run by a specific user (assuming that you have authority to do so) with `killall -u username`. See the `killall` man page for more options.

Using Priority Scheduling and Control

Two useful applications included with Ubuntu are the `nice` and `renice` commands. They are covered in Chapter 12, “Command-Line Master Class, Part 2.” Along with `nice`, system administrators can also use the `time` command to get an idea of how much time and what proportion of a system’s resources are required for a task, such as a shell script. (Here, `time` is used to measure the duration of elapsed time; the command that deals with civil and sidereal time is the `date` command.) This command is used with the name of another command (or script) as an argument, like this:

```
matthew@seymour:~$ sudo time -p find / -name conky
/home/matthew/conky
/etc/conky
/usr/lib/conky

/usr/bin/conky
real 30.19
user 1.09
sys 2.77
```

Output of the command displays the time from start to finish, along with the user and system time required. Other factors you can query include memory, CPU usage, and file system *input/output (I/O)* statistics. See the `time` command’s man page for more details.

The `top` command is covered in Chapter 12. It has some even-more-powerful cousins worth mentioning here.

One option for monitoring resource usage is called `htop`. It is not installed by default but is available from the Ubuntu software repositories and is worth a minute or two of your consideration when you’re familiar with `top`. Here are some key differences:

- ▶ With `htop`, you can scroll the list vertically and horizontally to see all processes and complete command lines.
- ▶ With `top`, you are subject to a delay for each unassigned key you press (which is especially annoying when multikey escape sequences are triggered by accident).
- ▶ `htop` starts faster. (`top` seems to collect data for a while before displaying anything.)
- ▶ With `htop`, you don’t need to type the process number to kill a process; with `top`, you do.
- ▶ With `htop`, you don’t need to type the process number or the priority value to `renice` a process; with `top`, you do.
- ▶ `htop` supports mouse operation; `top` doesn’t.
- ▶ `top` is older and therefore more used and tested.

See <https://hisham.hm/htop/> for more details.

Displaying Free and Used Memory with `free`

Although `top` includes some memory information, the `free` utility displays the amount of free and used memory in the system, in kilobytes. (The `-m` switch causes it to display in megabytes.) On one system, the output looks like this:

```
matthew@seymour:~$ free
              total        used        free      shared    buffers     cached
Mem:          4055680      3327764      727916           0       280944      2097568
-/+ buffers/cache:      949252      3106428
Swap:          8787512           0       8787512
```

This output describes a machine with 4GB of RAM memory and a swap partition of 8GB. Note that none of the swap is being used and that the machine is not heavily loaded. Linux is very good at memory management and “grabs” all the memory it can in anticipation of future work.

TIP

A useful trick is to employ the `watch` command, which repeatedly reruns a command every two seconds by default. If you use the following, you can see the output of the `free` command updated every two seconds:

```
matthew@seymour:~$ watch free
Use Ctrl+C to quit.
```

Another useful system-monitoring tool is `vmstat` (*virtual memory statistics*). This command reports on processes, memory, I/O, and CPU, typically providing an average since the last reboot; or you can make it report usage for a current period by telling it the time interval, in seconds, and the number of iterations you desire, like this:

```
matthew@seymour:~$ vmstat 5 10
```

This causes `vmstat` to run every five seconds for 10 iterations.

Use the `uptime` command to see how long it has been since the last reboot and to get an idea of what the load average has been; higher numbers mean higher loads.

Disk Space

Along with system load, it is important to keep an eye on the amount of free hard drive space that your computer has remaining. It is easy to do this, mainly by using the `df` command, as follows:

```
matthew@seymour:~$ df
```

Just using the command alone returns this output:

```
Filesystem      1K-blocks      Used Available Use% Mounted on
/dev/sda1        14421344    6584528    7104256  49% /
none              2020124         348    2019776   1% /dev
```

```

none                2027840      2456   2025384    1% /dev/shm
none                2027840        220   2027620    1% /var/run
none                2027840         0   2027840    0% /var/lock
none                2027840         0   2027840    0% /lib/init/rw
/dev/sda6           284593052 147323812 122812752 55% /home

```

Here you can see each drive as mounted on your system, as well as the used space, the available space, the percentage of the total usage of the disk, and where it is mounted on your system.

Unless you are good at doing math in your head, you might find it difficult to work out exactly what the figures mean in megabytes and gigabytes, so it is recommended that you use the `-h` switch to make the output human readable, like this:

```

matthew@seymour:~$ df -h
Filesystem      Size  Used Avail Use% Mounted on
/dev/sda1       14G  6.3G  6.8G  49% /
none            2.0G  348K  2.0G   1% /dev
none            2.0G  2.4M  2.0G   1% /dev/shm
none            2.0G  220K  2.0G   1% /var/run
none            2.0G    0   2.0G   0% /var/lock
none            2.0G    0   2.0G   0% /lib/init/rw
/dev/sda6       272G  141G  118G  55% /home

```

Disk Quotas

Disk quotas enable you to restrict the usage of disk space either by users or by groups. Although rarely—if ever—used on a local or standalone workstation, quotas are definitely a way of life at the enterprise level of computing. Usage limits on disk space not only conserve resources but also provide a measure of operational safety by limiting the amount of disk space any user can consume.

Disk quotas are more fully covered in Chapter 13, “Managing Users.”

Checking Log Files

Many of the services and programs that run on your computer save data in log files. Typical data include success and error messages for processes that are attempted and lists of actions. Some of these log files are extremely technical, whereas others are easily read and parsed by regular users who know what they are looking for. Most log files can be found in `/var/log/` or its subdirectories.

Typically, log files are used to learn about something that happened recently, so most admins are interested in the most recent entries. In this case, using `tail` is commonly used to read just the most recent 10 lines:

```

matthew@seymour:~$ tail /var/log/boot.log
* Starting                               [ OK ] *
Starting save kernel messages           [ OK ] * Starting

```

```

[ OK ] * Starting [ OK ] *
Starting deferred execution scheduler [ OK ] *
Starting regular background program processing daemon [ OK ] * Stopping
save kernel messages [ OK ] * Stopping
anac(h)ronistic cron [ OK ] * Starting CUPS
printing spooler/server [ OK ] * Starting CPU interrupts
balancing daemon [ OK ]

```

There isn't anything terribly interesting in this quote of today's `boot.log` on this machine, but it is sufficient to show how reading the last few lines of a log file works.

You are more likely to want to be able to find out whether something specific is mentioned in a log. The following example shows how to use `cat` and `grep` to look for mentions of `pnp` in `dmesg`, the display message buffer log for the Linux kernel, to see if there is any mention of a plug-and-play device:

```
matthew@seymour:~$ cat /var/log/dmesg | grep pnp
```

```

[ 0.426212] pnp: PnP ACPI init[ 0.426223] ACPI: bus type pnp registered[
0.426303] pnp 00:01: [dma 4][ 0.426315] pnp 00:01: Plug and Play ACPI device, IDs
PNP0200 (active)[ 0.426338] pnp 00:02: Plug and Play ACPI device, IDs PNP0b00
(active)[ 0.426351] pnp 00:03: Plug and Play ACPI device, IDs PNP0800 (active)[
0.426369] pnp 00:04: Plug and Play ACPI device, IDs PNP0c04 (active)[ 0.426531] pnp
00:05: [dma 0 disabled][ 0.426568] pnp 00:05: Plug and Play ACPI device, IDs PNP0501
(active)[ 0.426872] pnp 00:08: Plug and Play ACPI device, IDs PNP0103 (active)[
0.427298] pnp: PnP ACPI: found 12 devices[ 0.427299] ACPI: ACPI bus type pnp
unregistered

```

Here are a few of the most commonly used log files. Your system will have many others, in addition to these:

- ▶ `/var/log/apport.log`—Saves information about system crashes and reports
- ▶ `/var/log/auth.log`—Saves information about system access and authentication, including when a user does something using `sudo`
- ▶ `/var/log/kern.log`—Saves information from kernel messages, such as warnings and errors
- ▶ `/var/log/syslog`—Saves information from system events
- ▶ `/var/log/ufw.log`—Saves information from the Ubuntu Firewall
- ▶ `/var/log/apt/history.log`—Saves information about package installation and removal

Notice that the last one is in its own subdirectory. Many applications create their own directories and may even create multiple log files within their directories.

A couple of special cases deserve separate mention. These two are not read using standard methods, but each has its own program for reading from the command line. The commands are the same as the log names:

- ▶ **faillog**—Reads from `/var/log/faillog` and lists recent login failures
- ▶ **lastlog**—Reads from `/var/log/lastlog` and lists the most recent login for each account

For those who love GUI applications, there is a log reader installed by default in Ubuntu. Find System Log in the Dash to run it. It does not include every log in `/var/log`, but it does include the most important ones that serve the widest audience.

Rotating Log Files

Log files are great, but sometimes they can get unwieldy as time passes and more information is logged. Rotating log files prevents that problem. Rotating a log file means archiving the current log file, starting a fresh log, and deleting older log files. This means you always have a current log file to peruse (the previous log file), and the log files never grow too large.

Typically, log rotation is set up by an administrator to happen nightly, at a time when the system is not being heavily used. This is done with a utility called `logrotate` running as a `cron` job. (`cron` is described in Chapter 14, “Automating Tasks and Shell Scripting.”)

Ubuntu comes with `logrotate` installed. There is a `cron` job already set as well. You can find the script at `/etc/cron.daily/logrotate`. This file is a bash script and looks like this:

```
#!/bin/sh

# Clean nonexistent log file entries from status filecd /var/lib/logrotatetest -e
status || touch statushead -1 status > status.cleansed 's//g' status | while read
logfile datedo [ -e "$logfile" ] && echo "\"$logfile\" $date"done >> status.
cleanmv
status.clean status
test -x /usr/sbin/logrotate || exit 0/usr/sbin/logrotate /etc/logrotate.conf
```

Don't worry if you don't yet understand everything in this script. You don't need to understand it to configure `logrotate` to do what you want. You can learn more about bash and shell scripting in Chapter 14.

The important line right now is that last one, which lists the location of the configuration file for `logrotate`. Here are the default contents of `/etc/logrotate.conf`.

```
# see "man logrotate" for details

# rotate log files weekly
```



```

weekly
# use the syslog group by default, since this is the owning group# of /var/log/
syslog.su root syslog
# keep 4 weeks' worth of backlogsrotate 4
# create new (empty) log files after rotating old onescreate

# uncomment this if you want your log files compressed#compress
# packages drop log rotation information into this directoryinclude /etc/logrotate.d
# no packages own wtmp, or btmp -- we'll rotate them here/var/log/wtmp { missingok
monthly create 0664 root utmp rotate 1}
/var/log/btmp { missingok monthly create 0660 root utmp rotate 1}
# system-specific logs may be configured here

```

This file includes useful comments, and what it can configure is straightforward. If you can read the file, you probably already have a pretty accurate guess as to what the various settings do. As the first comment in the file says, the man page for `logrotate` has an explanation of everything that you can consult if it is not already clear.

One interesting entry says that packages drop log rotation information into `/etc/logrotate.d`. This is worth a closer look. The directory contains a config file for applications that are installed using the package manager and that log information. These files are named after the applications whose log files they control. Let's look at two examples. This first one is for `apt`, the package manager:

```

/var/log/apt/term.log { rotate 12 monthly compress missingok notifempty}
/var/log/apt/history.log { rotate 12 monthly compress missingok notifempty}

```

There are two entries here, each for a different log file that is created and used by `apt`. The entries define how many old versions of the log files to keep, how frequently `logrotate` will rotate the logs, whether to compress the log files, whether it is okay for the log file to be missing, and whether to bother rotating if the file is empty. Again, this is pretty straightforward. Here is a more complex example, for `rsyslog`, the system logging program:

```

/var/log/syslog
{
    rotate 7
    daily
    missingok
    notifempty
    delaycompress
    compress
    postrotate
        reload rsyslog >/dev/null 2>&1 || true
    endscript
}

```

```

/var/log/mail.info
/var/log/mail.warn
/var/log/mail.err
/var/log/mail.log
/var/log/daemon.log
/var/log/kern.log
/var/log/auth.log
/var/log/user.log
/var/log/lpr.log
/var/log/cron.log
/var/log/debug
/var/log/messages
{
    rotate 4
    weekly
    missingok
    notifempty
    compress
    delaycompress
    sharedscripts
    postrotate
        reload rsyslog >/dev/null 2>&1 || true
    endscript
}

```

The man page for `logrotate` defines all the commands used in these configuration files, but many are probably clear to you already. Here are some of the most important ones:

- ▶ **rotate**—Defines how many archived logs are kept at any one time
- ▶ **interval**—Defines how often to rotate the log; the actual setting will be daily, weekly, monthly, or yearly
- ▶ **size**—Defines how large a log file can become before it is rotated; this setting supercedes the preceding time interval setting, and the format will be a number and a unit, such as `size 512k` OR `size 128M` OR `size 100G`
- ▶ **compress**—Configures the log file to be compressed
- ▶ **nocompress**—Configures the log file not to be compressed

What is more important to cover here than all the individual options, which you can look up in the man page, is that these individual configuration files for specific applications will override the default settings in `/etc/logrotate.conf`. If a setting is assigned a value in that file, it will be used by all applications that `logrotate` affects unless an application-specific file in `/etc/logrotate.d` includes the same setting.

Graphical Process- and System-Management Tools

The GNOME and KDE desktop environments offer a rich set of network and system-monitoring tools. Graphical interface elements, such as menus and buttons, and graphical output, including metering and real-time load charts, make these tools easy to use. These clients, which require an active X session and in some cases root permission, are included with Ubuntu.

If you view the graphical tools locally while they are being run on a server, you must have X properly installed and configured on your local machine. Although some tools can be used to remotely monitor systems or locally mounted remote file systems, you have to properly configure pertinent X11 environment variables, such as `$DISPLAY`, to use the software or use the `ssh` client's `-x` option when connecting to the remote host.

System Monitor

System Monitor is a graphical monitoring tool that is informative, easy to use and understand, and very useful. It has tabs for information about running processes, available resources, and local file systems.

Conky

Conky is a highly configurable, rather complex system monitor that is light on system resources and can give you information about nearly anything. The downside is that you need to learn how to configure it. Simply installing Conky from the software repositories only gets you started. However, for those who want specific information displayed on their desktop at all times, it is invaluable and well worth the time it takes to figure it out. We give an example here, but to truly appreciate the power, flexibility, and possibilities of Conky, visit <https://github.com/brndnmtthws/conky>.

Conky uses text files for configuration and is often started using a short script. The example shown in Figure 16.1 is from Matthew's personal configuration on his workstation and is intended as a simple example to get you started.

In this example, Conky gives information about the kernel, the operating system, the hostname of the system, and the current time and date. It continually updates with information on load averages, CPU usage and temperature, battery status, and RAM and disk usage. In addition, it shows networking details, including the internal network IP address and the IP address assigned to the outside-facing router the network uses to connect to the wider world and current inbound and outbound network connections. (The IP address is assigned by the ISP, and it changes, so if you try to attack Matthew's home network using it, you will find that the IP address is being used by someone else now.) That is a lot of information in a small space. This setup is not as pretty as some you will see at the previous links, nearly all of which have their setup details made freely available by the person who wrote the configurations.


```

-- Xft font when Xft is enabled
--font = 'HackBold:size=9',
    font = 'Ubuntu:size=8',

-- Text alignment, other possible values are commented
--minimum_size 10 10
    gap_x = 13,
    gap_y = 45,
--alignment top_left
    alignment = 'top_right',
--alignment bottom_left
--alignment bottom_right

-- Add spaces to keep things from moving about? This only affects certain objects.
    use_spacer = 'right',

-- Subtract file system buffers from used memory?
    no_buffers = true,

-- Use double buffering (reduces flicker, may not work for everyone)
    double_buffer = true,

-- Allows icons to appear, window to be moved, and transparency
    own_window = true,
    own_window_class = 'Conky',--new
--own_window_type override
    own_window_transparent = true,
--own_window_hints undecorated,below,skip_taskbar
    own_window_type = 'normal',--new
    own_window_hints = 'undecorated,below,sticky,skip_taskbar',--new
    own_window_argb_visual = true,--new
    own_window_argb_value = 0,--new

-- set temperature units, either "fahrenheit" or "celsius"
    temperature_unit = "fahrenheit",

-- set to yes if you want Conky to be forked in the background
    background = true,

-- Update interval in seconds
    update_interval = 1,

    cpu_avg_samples = 1,
    net_avg_samples = 1,

-- ----- start display config -----

```

```

};

conky.text = [[
${alignc}${color c68be8}$sysname kernel $kernel
${alignc}${color c68be8}${exec cat /etc/issue.net} on $machine host $nodename

${color c68be8}Time:${color E7E7E7}    $time
${color c68be8}Updates: ${color E7E7E7}${execi 3600 /usr/lib/update-
[ccc]notifier/apt _check.py --human-readable | grep updated}
${color c68be8}Security: ${color E7E7E7}${execi 3600 /usr/lib/update-
[ccc]notifier/apt _check.py --human-readable | grep security}

${color c68be8}Disk Usage:
  Samsung SSD 850 PRO 256GB: ${color E7E7E7}/
    ${fs _used /}/${fs _size /} ${fs _bar /}
  ${color c68be8}Crucial _CT1050MX300SSD1 1.1TB: ${color E7E7E7}/home
    ${fs _used /home}/${fs _size /home} ${fs _bar /home}
  ${color c68be8}Hitachi Ultrastar A7K2000 HUA722010CLA330 1TB: ${color E7E7E7}/home/matt/
Media
    ${fs _used /home/matt/Media}/${fs _size /home/matt/Media} ${fs _bar /home/matt/Media}
  ${color c68be8}Seagate Expansion Disk (External) 5TB: ${color E7E7E7}/media/matt/Backups
    ${fs _used /media/matt/Backups}/${fs _size /media/matt/Backups} ${fs _bar /media/matt/
Backups}

${color c68be8}RAM Usage:${color E7E7E7} $mem/$memmax - $memperc% $membar
${color c68be8}Swap Usage:${color E7E7E7} $swap/$swapmax - $swapperc% ${swapbar}

${color c68be8}Load average:${color E7E7E7}    $loadavg
${color c68be8}Processes:${color E7E7E7} $processes  ${color c68be8}Running:${color E7E7E7}
$running _processes ${color c68be8}

${color c68be8}CPU usage          ${alignr}PID      CPU%   MEM%
${color E7E7E7} ${top name 1}${alignr}${top pid 1}    ${top cpu 1}  ${top mem 1}
${color E7E7E7} ${top name 2}${alignr}${top pid 2}    ${top cpu 2}  ${top mem 2}
${color E7E7E7} ${top name 3}${alignr}${top pid 3}    ${top cpu 3}  ${top mem 3}

${color c68be8}Memory usage
${color E7E7E7} ${top _mem name 1}${alignr}${top _mem pid 1}  ${top _mem cpu 1}
${top _mem mem 1}
${color E7E7E7} ${top _mem name 2}${alignr}${top _mem pid 2}  ${top _mem cpu 2}
${top _mem mem 2}
${color E7E7E7} ${top _mem name 3}${alignr}${top _mem pid 3}  ${top _mem cpu 3}
${top _mem mem 3}

```

```

${color c68be8}Current CPU usage:
  ${color c68be8}CPU0:${color E7E7E7}  ${cpu cpu0}%  ${color c68be8}CPU1:${color E7E7E7}
${cpu cpu1}%  ${color c68be8}CPU2:${color E7E7E7}  ${cpu cpu2}%  ${color c68be8}
CPU3:${color E7E7E7}  ${cpu cpu3}%
  ${color c68be8}CPU4:${color E7E7E7}  ${cpu cpu4}%  ${color c68be8}CPU5:${color E7E7E7}
${cpu cpu5}%  ${color c68be8}CPU6:${color E7E7E7}  ${cpu cpu6}%  ${color c68be8}
CPU7:${color E7E7E7}  ${cpu cpu7}%
  ${color c68be8}CPU8:${color E7E7E7}  ${cpu cpu8}%  ${color c68be8}CPU9:${color E7E7E7}
${cpu cpu9}%  ${color c68be8}CPU10:${color E7E7E7}  ${cpu cpu10}%  ${color c68be8}
CPU11:${color E7E7E7}  ${cpu cpu11}%
  ${color c68be8}CPU12:${color E7E7E7}  ${cpu cpu12}%  ${color c68be8}CPU13:${color
E7E7E7}  ${cpu cpu13}%  ${color c68be8}CPU14:${color E7E7E7}  ${cpu cpu14}%  ${color c68be8}
CPU15:${color E7E7E7}  ${cpu cpu15}%
  ${color c68be8}CPU16:${color E7E7E7}  ${cpu cpu16}%  ${color c68be8}CPU17:${color E7E7E7}
${cpu cpu17}%  ${color c68be8}CPU18:${color E7E7E7}  ${cpu cpu18}%  ${color c68be8}
CPU19:${color E7E7E7}  ${cpu cpu19}%
  ${color c68be8}CPU20:${color E7E7E7}  ${cpu cpu20}%  ${color c68be8}CPU21:${color E7E7E7}
${cpu cpu21}%  ${color c68be8}CPU22:${color E7E7E7}  ${cpu cpu22}%  ${color c68be8}
CPU23:${color E7E7E7}  ${cpu cpu23}%

```

```

${color c68be8}Current CPU core temps (cores with sensors):
${color E7E7E7}${execi 2 sensors | grep "Core" | cut -c 1-22}

```

```

${color c68be8}Wired Networking:
  ${color c68be8}Local IP:  ${color E7E7E7}${addr enp1s0}  ${color c68be8}
  ${color c68be8}total download:  ${color E7E7E7}${totaldown enp1s0}
  ${color c68be8}total upload:  ${color E7E7E7}${totalup enp1s0}
  ${color c68be8}download speed:  ${color E7E7E7}${downspeed enp1s0}${color E7E7E7}  ${color
c68be8} upload speed:  ${color E7E7E7}${upspeed enp1s0}
  ${color E7E7E7}${downspeedgraph enp1s0 15,150 E7E7E7 DA9347}  ${alignr}${color
E7E7E7}${upspeedgraph enp1s0 15,150 DA9347 E7E7E7}

```

```

${color c68be8}Public IP:  ${color E7E7E7}${execi 600 bash /home/matt/conky/myip.sh}

```

```

${color c68be8}Port(s) / Connections:
${color c68be8}Inbound:  ${color E7E7E7}${tcp_portmon 1 32767 count}  ${color c68be8}Out-
bound:  ${color E7E7E7}${tcp_portmon 32768 61000 count}  ${color c68be8}Total:  ${color
E7E7E7}${tcp_portmon 1 65535 count}
${color c68be8}Outbound Connection  ${alignr} Remote Service/Port${color E7E7E7}
  ${tcp_portmon 1 65535 rhost 0}  ${alignr}  ${tcp_portmon 1 65535 rservice 0}
  ${tcp_portmon 1 65535 rhost 1}  ${alignr}  ${tcp_portmon 1 65535 rservice 1}
  ${tcp_portmon 1 65535 rhost 2}  ${alignr}  ${tcp_portmon 1 65535 rservice 2}
  ${tcp_portmon 1 65535 rhost 3}  ${alignr}  ${tcp_portmon 1 65535 rservice 3}
  ${tcp_portmon 1 65535 rhost 4}  ${alignr}  ${tcp_portmon 1 65535 rservice 4}
  ${tcp_portmon 1 65535 rhost 5}  ${alignr}  ${tcp_portmon 1 65535 rservice 5}
]];

```

Most of these details are clear, but one is particularly interesting. There are commercial and other sites that, if you visit them, will return your IP address. This is easily accomplished several ways. Matthew chose to put the following PHP in a file named `myip.php` on a server he owns, and he calls it directly:

```
<?
$remote = $_SERVER["REMOTE_ADDR"];
    echo $remote;
?>
```

Doing this can help you not feel guilty about constantly hitting someone else's server for this information.

Finally, although you can run Conky from the command line at any time after it is set up, to make it more convenient, many people choose to keep their config files in their `/home` directory somewhere and then write a script with the custom location. If you add a pause to the beginning of the script, you can add the script to Startup Applications and have it come up after all your other desktop processes are up and running. Here is a simple example:

```
#!/bin/bash
sleep 45 &&
exec conky -d -c ~/conky/conkyrc &
exit
```

Save it in `/home/username/conky` along with all your Conky config files, make it executable, and then have the Startup Applications process call it at bootup. Note that this way, you can also run more than one instance of Conky at a time, perhaps having your regular instance in the upper right of the screen and a weather instance or something else in a different location. The possibilities are vast.

A lovely GUI program for creating and managing Conky configurations exists, but it is not in the Ubuntu software repositories. If you are interested in exploring it further, there is a PPA from which you can install Conky Manager at <https://launchpad.net/conky-manager>. Installation directions and other documentation are available from the maintainer's website, at www.teejeetech.in/p/conky-manager.html.

Other Graphical Process- and System-Monitoring Tools

Graphical system- and process-monitoring tools that are available for Ubuntu include the following:

- ▶ **vncviewer**—AT&T's open-source remote session manager (part of the Xvnc package), which you can use to view and run a remote desktop session locally. This software (discussed in more detail in Chapter 19, "Remote Access with SSH and VNC") requires an active background X session on the remote computer.
- ▶ **gnome-nettool**—A GNOME-developed tool that enables system administrators to carry out a wide range of diagnostics on network interfaces, including port scanning and route tracing.

- ▶ **wireshark**—A graphical network protocol analyzer that can be used to save or display packet data in real time and that has intelligent filtering to recognize data signatures or patterns from a variety of hardware and data captures from third-party data-capture programs, including compressed files. Some protocols include Internet Protocol (IP), Transmission Control Protocol (TCP), Domain Name System (DNS), Hypertext Transfer Protocol (HTTP), Secured Shell (SSH), Transport Layer Security (TLS), and Hypertext Transfer Protocol over SSL/TLS (HTTPS).

KDE Process- and System-Monitoring Tools

KDE provides several process- and system-monitoring clients. Integrate the KDE graphical clients into the desktop taskbar by right-clicking the taskbar and following the menus.

These KDE monitoring clients include the following:

- ▶ **kdf**—A graphical interface to your system's file system table that displays free disk space and enables you to mount and unmount file systems with a pointing device
- ▶ **ksysguard**—Another panel applet that provides CPU load and memory use information in animated graphs

Enterprise Server Monitoring

Servers used in enterprise situations require extreme uptime. Monitoring is vital to know how your system is performing and to have observability into the general health of your system components and services.

It is beyond the scope of this book to discuss topics such as observability, redundancy, failsafe and failover safeguards, and so on in any depth. However, the tools listed in this section can help enterprise sysadmins get started in their search for the proper tool(s) for their setting.

Datadog

Datadog is one of several young monitoring companies that have designed their products as software as a service (SaaS) cloud-native applications. Datadog requires you to install an agent in your system. You can do the rest using their web UI (an API is available to also programmatically access their platform, if you prefer). Then, you learn about the various integrations and decide which metrics you want to track, and finally you create dashboards, graphs, and monitors. It is proprietary and has multiple pricing options depending on your needs.

Nagios

Nagios has an open source foundation and some core monitoring options that are free to download, use, and modify. The paid option offers far more features and flexibility. In both cases, you must download the Nagios software and install on dedicated resources within your IT infrastructure. Nagios has been at the forefront of monitoring for years but is gradually giving way to SaaS newcomers in the new cloud computing era.

New Relic

New Relic is another of the several young monitoring companies that have designed their products as software as a service (SaaS) cloud-native applications. It is flexible, configurable, and designed to work across your cloud deployments. New Relic's products are proprietary, and the company offers multiple pricing options depending on your needs.

SignalFx

SignalFx is another of the several young monitoring companies that have designed their products as software as a service (SaaS) cloud-native applications. It is flexible, configurable, and designed to work across your cloud deployments. SignalFx's products are proprietary, and the company offers multiple pricing options depending on your needs. SignalFx was recently bought by Splunk, who is a long-time player in the on-site monitoring business and wants to stay relevant with the transition to cloud computing.

Splunk

Once a big player in enterprise monitoring, Splunk has had to shift course of late as enterprises have been moving to cloud computing, and they have faced strong competition from the smaller, young monitoring companies already mentioned. They are still, with Nagios, one of the mainstays of on-premise data analytics solutions. Splunk is the standard for some unique features like security information and event management (SIEM), artificial intelligence for IT operations (AIOps), and compliance monitoring.

Zabbix

Zabbix is more popular in Europe than in North America. Similar to Splunk, it is a mature enterprise platform designed to monitor large-scale IT environments, specifically on-premise, but the company has broadened its abilities and works across all the various popular cloud platforms. It is free and open source, with paid support, services, and training available.

References

- ▶ <https://and.sourceforge.net>—Home page of the *auto nice daemon* (AND), which can be used to prioritize and reschedule processes automatically
- ▶ <https://sourceforge.net/projects/schedutils/>—Home page for various projects offering scheduling utilities for real-time scheduling
- ▶ www.datadoghq.com/—Home page for Datadog
- ▶ www.nagios.com/—Home page for Nagios
- ▶ <https://newrelic.com/>—Home page for New Relic
- ▶ www.signalfx.com/—Home page for SignalFx
- ▶ www.splunk.com/—Home page for Solarwinds
- ▶ <https://sourceforge.net/projects/schedutils/>—Home page for Zabbix

This page intentionally left blank

CHAPTER 17

Backing Up

This chapter examines the practice of safeguarding data by creating backup copies, restoring that same data if necessary, and recovering data in case of a catastrophic hardware or software failure. The chapter gives you a full understanding of the reasons for sound backup practices. You can use the information here to make intelligent choices about which strategies are best for you. The chapter also shows you how to perform some types of data recovery and system restoration on your own and provides advice about when to seek professional assistance.

Choosing a Backup Strategy

Backups are always trade-offs. Any backup consumes time, money, and effort on an ongoing basis; backups must be monitored, validated, indexed, and stored, and you must continuously purchase new media. Sound expensive? The cost of not having backups is the loss of your critical data. Re-creating data from scratch costs time and money, and if the cost of doing it all again is greater than the cost associated with backing up, you should be performing backups. At their most basic, backups are insurance against financial loss for you or your business.

Your first step in formulating and learning to use an effective backup strategy is to choose the strategy that is right for you. First, you must understand some of the most common (and not-so-common) causes of data loss so that you can better understand the threats your system faces. Then, you need to assess your own system, how it is used and by whom, your available hardware and software resources, and your budget constraints. The following sections look at each of these issues in detail and provide some backup system examples.

IN THIS CHAPTER

- ▶ Choosing a Backup Strategy
- ▶ Choosing Backup Hardware and Media
- ▶ Using Backup Software
- ▶ Copying Files
- ▶ Version Control for Configuration Files
- ▶ System Rescue
- ▶ References

Why Data Loss Occurs

Files may disappear for any number of reasons: They can be lost because hardware fails and causes data loss; or if your attention wanders, you might accidentally delete or overwrite a file. Some data loss occurs as a result of natural disasters and other circumstances beyond your control. A tornado, a flood, or an earthquake could strike; the water pipes could burst; or the building could catch on fire. Your data, as well as the hardware, would likely be destroyed in such a disaster. A disgruntled employee might destroy files or hardware in an attempt at retribution. Equipment can be stolen, or it might fail; all equipment fails at some time—most likely when it is extremely important for it not to fail.

A CASE IN POINT

A recent Harris poll of Fortune 500 executives found that roughly two-thirds of them had problems with their backup and disaster recovery plans. How about you?

Data can also be lost because of malfunctions that corrupt the data as it is being written to the disk. Other applications, utilities, and drivers might be poorly written, buggy (the phrase most often heard today is “still beta quality”), or might suffer some corruption and fail to correctly write that all-important data you have just created. If that happened, the contents of your data file would be indecipherable garbage of no use to anyone.

All these accidents and other disasters offer important reasons for having a good backup strategy; however, the most frequent cause of data loss is human error. Who among us has not overwritten a new file with an older version or unintentionally deleted a needed file? This applies not only to data files, but also to configuration files and binaries. In mail lists, Usenet newsgroup postings, and online forums, stories about deleting entire directories such as `/home`, `/usr`, or `/lib` are all too common. On a stable server that is not frequently modified or updated, you can choose to mount `/usr` read-only to prevent writing over or deleting anything in it. Incorrectly changing a configuration file and not saving the original in case it has to be restored (which happens more often than not because the person reconfigured it incorrectly) is another common error.

TIP

To make a backup of a configuration file you are about to edit, use the `cp` command:

```
matthew@seymour:~$ cp filename filename.original
```

To restore it, use the following:

```
matthew@seymour:~$ cp filename.original filename
```

Never edit or move the `*.original` file, or the original copy will be lost. You can change the file's mode to be unwritable; then if you try to delete it, you are prevented from doing so and receive a warning.

Proper backups can help you recover from these problems with a minimum of hassle, but you have to put in the effort to keep backups current, verify that they are intact, and practice restoring the data in different disaster scenarios.

Assessing Your Backup Needs and Resources

By now you have realized that some kind of plan is needed to safeguard your data, and, like most others, you may be overwhelmed by the prospect. Entire books, as well as countless articles and white papers, have been written on the subject of backing up and restoring data. What makes the topic so complex is that each solution is truly individual. However, the proper approach to making the decision is very straightforward. You start the process by answering two questions:

- ▶ What data must be safeguarded?
- ▶ How often does the data change?

The answers to these two questions help you determine how important the data is, understand the volume of the data, and determine the frequency of the backups. This information, in turn, helps you choose the backup medium. Only then can you select the software to accommodate all these considerations. (You learn about choosing backup software, hardware, and media later in this chapter.)

Available resources are another important consideration when selecting a backup strategy. Backups require time, money, and personnel. Begin your planning activities by determining what limitations you face for all these resources. Then construct your plan to fit those limitations—or be prepared to justify the need for more resources with a careful assessment of both backup needs and costs.

TIP

If you are not willing or able to assess your backup needs and choose a backup solution, you can choose from the legions of consultants, hardware vendors, and software vendors that are available to assist you. The best way to choose one in your area is to ask other UNIX and Linux system administrators (located through user groups, discussion groups, or mail lists) who are willing to share their experiences and make recommendations. If you cannot get a referral, ask consultants for references and check them out.

Many people also fail to consider the element of time when formulating backup plans. Some backup devices are faster than others, and some recovery methods are faster than others. You need to consider that when making choices.

To formulate a backup plan, you need to determine the frequency of backups. The necessary frequency of backups should be determined by how quickly the important data on your system changes. On a home system, most files never change, a few change daily, and some change weekly. No elaborate strategy needs to be created to deal with that. A good strategy for home use is to back up (to any kind of removable media) critical data frequently and back up configuration and other files weekly.

At the enterprise level on a larger system with multiple users, a different approach is called for. Some critical data changes constantly, and it could be expensive to re-create this data because doing so typically involves elaborate and expensive solutions. Most of us exist somewhere in between these extremes. Assess your system and its use to determine where you fall in this spectrum.

Backup schemes and hardware can be elaborate or simple, but they all require a workable plan and faithful follow-through. Even the best backup plan is useless if the process is not carried out, data is not verified, and data restoration is not practiced on a regular basis. Whatever backup scheme you choose, be sure to incorporate in it these three principles:

- ▶ **Have a plan**—Design a plan that is right for your needs and have equipment appropriate to the task. This involves assessing all the factors that affect the data you are backing up. We delve into more detail later in the chapter.
- ▶ **Follow the plan**—Faithfully complete each part of your backup strategy and verify the data stored in the backups. Backups with corrupt data are of no use to anyone. Even backup operations can go wrong.
- ▶ **Practice your skills**—Practice restoring data from your backup systems from time to time so that when disaster strikes, you are ready (and able) to benefit from the strength of your backup plan. (For restoring data, see the section “Using Backup Software.”) Keep in mind that it is entirely possible that the flaws in your backup plan will become apparent only when you try restoring.

SOUND PRACTICES

You have to create your own best backup plan, but here are some building blocks that go into the foundation of any sound backup program:

- ▶ Maintain more than one copy of critical data.
- ▶ Label backups.
- ▶ Store backups in a climate-controlled and secure area.
- ▶ Use secure offsite storage of critical data. Many companies choose bank vaults for their offsite storage, and this is highly recommended.
- ▶ Establish a backup policy that makes sense and can be followed religiously. Try to back up your data when the system is consistent (that is, no data is being written), which is usually overnight.
- ▶ Keep track of who has access to your backup media and keep the total number of people as low as possible. If you can, allow only trusted personnel near your backups.
- ▶ Routinely verify backups and practice restoring data from them.
- ▶ Routinely inspect backup media for defects and regularly replace them (after destroying the data on them if it is sensitive).

Evaluating Backup Strategies

When you are convinced that you need backups, you need a strategy. Being specific about an ideal strategy is difficult because each user’s or administrator’s strategy will be highly individualized, but here are a few general examples:

- ▶ **Home user**—At home, the user has the Ubuntu installation media that takes less than an hour to reinstall, so the time issue is not a major concern. The home user will want to back up any configuration files that have been altered, keep an archive of any files that have been downloaded, and keep an archive of any data files created while using any applications. Unless the home user has a special project in which constant backups are useful, a weekly backup is probably adequate. The home user will likely use a consumer-focused online cloud service like Dropbox, an external hard drive, or other removable media for backups.
- ▶ **Small office**—Many small offices tend to use the same strategy as home users but are more likely to back up critical data daily and use an automated cloud service. Although they will use scripts to automate backups, some of this is probably still being done by hand.
- ▶ **Small enterprise**—Here is where backups begin to require higher-end equipment with fully automated on- and off-site backups. Commercial backup software usually makes an introduction at this level, but a skillful system administrator on a budget can use one of the basic applications discussed in this chapter. Backups are highly structured and supervised by a dedicated system administrator. You might have guessed that small enterprises are also moving their backups to online cloud services.
- ▶ **Large enterprise**—Large enterprises are the most likely candidates for the use of expensive, proprietary, highly automated backup solutions. At this level, data means money, lost data means lost money, and delays in restoring data means money lost as well. These system administrators know that backups are necessary insurance and plan accordingly. Often, they own their own online, distributed cloud systems, with multiple redundant data centers in geographically diverse locations.

Does all this mean that enterprise-level backups are better than those done by a home user? Not at all. The “little guy” with Ubuntu can do just as well as the enterprise operation by investing more time in the process. By examining the higher-end strategies in this chapter, therefore, we can apply useful concepts across the board.

This chapter focuses on local-level activities, not the cloud-services activities that are based on techniques like those listed here, but combined with networking and cloud-service-specific additional details. This chapter also discusses some technologies that are a bit outdated for the enterprise but might be useful to a hobbyist with cheap and easy access to older equipment. If you want to use an online cloud service, take what you learn here, read everything made available by your cloud service provider, and then do your homework to design a suitable backup solution for your unique needs. This could be as simple as putting all your important files in a Dropbox-style cloud folder that automatically updates to another computer you own. This can work well if you are a casual consumer-grade user backing up simple documents and a few media files and remember that services like these generally do not guarantee that your data will be permanently backed up (especially with the free versions). Although we’ve not had problems with these solutions, we warn that they are not enterprise backup solutions. You might need to study up on

Amazon Web Services, OpenStack, or other cloud providers and learn the fine details of their services to see if they suit your needs.

NOTE

If you are a new system administrator, you might inherit an existing backup strategy. Take some time to examine it and see if it meets the current needs of the organization. Think about what backup protection your organization really needs and determine whether the current strategy meets that need. If it does not, change the strategy. Consider whether the current policy is being practiced by the users, and, if not, why it is not.

BACKUP LEVELS

UNIX uses the concept of backup levels as a shorthand way of referring to how much data is backed up in relation to a previous backup. It works this way: A level 0 backup is a full backup. The next backup level is 1.

Backups at the other numbered levels will back up everything that has changed since the last backup at that level or a numerically higher level. (The `dump` command, for example, offers 10 different backup levels.) For example, a level 3 followed by a level 4 backup generates an incremental backup from the full backup, and a level 4 followed by a level 3 generates a differential backup between the two.

The following sections examine a few of the many strategies in use today. Many strategies are based on these sample schemes; one of them can serve as a foundation for the strategy you construct for your own system.

Simple Strategy

If you need to back up just a few configuration files and some small data files, copy them to a USB stick, label it, and keep it someplace safe. Most users have switched to using an external hard drive for backups because they are becoming less and less expensive and hold a great amount of data, or they have moved backups online.

In addition to configuration and data files, you should archive each user's `/home` directory and entire `/etc` directory. Between the two, that backup would contain most of the important files for a small system. Then, if necessary, you can easily restore this data from the backup media device you have chosen after a complete reinstall of Ubuntu.

Experts used to say that if you have more data than can fit on a floppy disk, you really need a formal backup strategy. Because a floppy disk only held a little over 1MB (and is now incredibly obsolete), perhaps we should change that to "if you have more data than can fit on a cheap USB stick." In any case, some formal backup strategies are discussed in the following sections.

Full Backup on a Periodic Basis

A full backup on a periodic basis is a strategy that involves a backup of the complete file system on a weekly, biweekly, or other periodic basis. The frequency of the backup

depends on the amount of data being backed up, the frequency of changes to the data, and the cost of losing those changes.

This backup strategy is not complicated to perform, and it can be accomplished with the swappable disk drives discussed later in the chapter. If you are connected to a network, it is possible to mirror the data on another machine (preferably offsite); the `rsync` tool is particularly well suited to this task. Recognize that this does not address the need for archives of the recent state of files; it only presents a snapshot of the system at the time the update is done.

Full Backups with Incremental Backups

Another scheme involves performing a full backup of the entire system once a week, along with a daily incremental backup of only those files that have changed in the previous day, and it begins to resemble what a system administrator of a medium to large system traditionally uses.

This backup scheme can be advanced in two ways. First, each incremental backup can be made with reference to the original full backup. In other words, a level 0 backup is followed by a series of level 1 backups. The benefit of this backup scheme is that a restoration requires only two tapes (the full backup and the most recent incremental backup). But because it references the full backup, each incremental backup might be large (and could grow ever larger) on a heavily used system.

Alternatively, each incremental backup could reference the previous incremental backup. This is a level 0 backup followed by a level 1, followed by a level 2, and so on. Incremental backups are quicker (less data each time) but require every tape to restore a full system. Again, it is a classic trade-off decision.

Modern commercial backup applications such as Amanda or BRU assist in organizing the process of managing complex backup schedules and tracking backup media. Doing it yourself using the classic `dump` or employing shell scripts to run `tar` requires that system administrators handle all the organization themselves. For this reason, complex backup situations are typically handled with commercial software and specialized hardware packaged, sold, and supported by vendors.

Mirroring Data or RAID Arrays

Given adequate (and often expensive) hardware resources, you can always mirror the data somewhere else, essentially maintaining a real-time copy of your data on hand. This is often a cheap, workable solution if no large amounts of data are involved. The use of *redundant array of independent disks (RAID)* arrays (in some of their incarnations) provides for recovery if a disk fails.

Note that RAID arrays and mirroring systems just as happily write corrupt data as valid data. Moreover, if a file is deleted, a RAID array will not save it. RAID arrays are best suited for protecting the current state of a running system, not for backup needs.

Making the Choice

Only you can decide what is best for your situation. After reading about the backup options in this book, put together some sample backup plans; then run through a few likely scenarios and assess the effectiveness of your choice.

In addition to all the other information you have learned about backup strategies, here are a couple good rules of thumb to remember when making your choice:

- ▶ If the backup strategy and policy is too complicated (and this holds true for most security issues), it will eventually be disregarded and fall into disuse.
- ▶ The best scheme is often a combination of strategies; use what works.

Choosing Backup Hardware and Media

Any device that can store data can be used to back it up, but that is like saying that anything with wheels can take you on a cross-country trip. Trying to fit 10GB worth of data on a big stack of CD-RWs or DVD-RWs is an exercise in frustration, and using an expensive automated tape device to save a single copy of an email is a waste of resources. In addition, those technologies are rapidly disappearing.

In this section, you find out about some of the most common backup hardware available and how to evaluate its appropriateness for your backup needs. With large storage devices becoming increasingly affordable (you can now get multiple-TB hard drives for around \$100), decisions about backup hardware for small businesses and home users have become more interesting.

External Hard Drive

This is an easy option. Buy an external hard drive that connects to your system via USB and copy important data to it regularly. This has replaced past recommendations for most home users.

Network Storage

For network backup storage, remote arrays of hard drives provide one solution to data storage. With the declining cost of mass storage devices and the increasing need for larger storage space, network storage (*NAS*, or *network-attached storage*) is available and supported in Linux. Network storage involves cabinets full of hard drives and their associated controlling circuitry, as well as special software to manage all of it. NAS systems are connected to the network and act as huge (and expensive) mass storage devices.

More modest and simple network storage can be done on a remote desktop-style machine that has adequate storage space (up to eight 1TB drives is a lot of storage space, easily accomplished with off-the-shelf parts), but then that machine and the local system administrator have to deal with all the problems of backing up, preserving, and restoring the data. Several hardware vendors offer such products in varying sizes.

Tape Drive Backups

While this is becoming less common, tape drive backup is a viable technology that is still in use. Tape drives have been used in the computer industry from the beginning. Tape drive storage has been so prevalent in the industry that the `tar` command (the most commonly used command for archiving) is derived from the words *tape archive*. Capacities and durability of tapes vary from type to type and range from a few gigabytes to hundreds of gigabytes, with commensurate increases in cost for the equipment and media. Autoloading tape-drive systems can accommodate archives that exceed the capacity of the file systems.

TIP

Older tape equipment is often available in the used equipment market and might be useful for smaller operations that have outgrown more limited backup device options.

Tape equipment is well supported in Linux and, when properly maintained, is extremely reliable. The tapes themselves are inexpensive, given their storage capacity and the ability to reuse them. Be aware, however, that tapes do deteriorate over time and, being mechanical, tape drives can and will fail.

CAUTION

Neglecting to clean, align, and maintain tape drives puts your data at risk. The tapes themselves are also susceptible to mechanical wear and degradation. Hardware maintenance is part of a good backup policy. Do not ever forget that it is a question of when—not if—hardware will fail.

Cloud Storage

Services such as Dropbox and Amazon's AWS and S3 offer a way to create and store backups offsite. Larger companies may create their own offsite, online storage options as well. In each of these and similar cases, data is copied and stored remotely on a file server set aside specifically for that purpose. The data backups may be scheduled with great flexibility and according to the plans and desires of the customer.

Cloud storage is a backup solution that is recent and growing in popularity, but it is also a technology that is changing rapidly. To learn more about the options mentioned here, take a look at www.dropbox.com and <https://aws.amazon.com/s3/>. Although these are not the only services of the kind available, they offer a good introduction to the concept. If you like to “roll your own,” you definitely want to take a look at Ubuntu Enterprise Cloud at www.ubuntu.com/cloud.

Using Backup Software

Because there are thousands of unique situations requiring as many unique backup solutions, it is not surprising that Linux offers many backup tools. Along with command-line

tools such as `tar` and `dd`, Ubuntu also provides a graphical archiving tool for desktop installations called *Déjà Dup* that is quite powerful. Another excellent but complicated alternative is the Amanda backup application—a sophisticated backup application that works well over network connections and can be configured to automatically back up all the computers on a network. Amanda works with drives as well as tapes.

NOTE

The software in a backup system must support the hardware, and this relationship can determine which hardware or software choices you make. Many system administrators choose particular backup software not because they prefer it to other options but because it supports the hardware they own.

The price seems right for free backup tools, but consider the software's ease of use and automation when assessing costs. If you must spend several hours implementing, debugging, documenting, and otherwise dealing with overly elaborate automation scripts, the real costs go up.

tar: The Most Basic Backup Tool

The `tar` tool, the bewhiskered old man of archiving utilities, is installed by default. It is an excellent tool for saving entire directories full of files. For example, here is the command used to back up the `/etc` directory:

```
matthew@seymour:~$ sudo tar cvf etc.tar /etc
```

This example uses `tar` to create an archive, calls for verbose message output, and uses the filename `etc.tar` as the archive name for the contents of the directory `/etc`.

Alternatively, if the output of `tar` is sent to the standard output and redirected to a file, the command appears as follows:

```
matthew@seymour:~$ sudo tar cv /etc > etc.tar
```

The result is the same as with the preceding `tar` options: All files in the `/etc` directory will be saved to a file named `etc.tar`.

With an impressive array of options (see the man page), `tar` is quite flexible and powerful in combination with shell scripts. With the `-z` option, it can even create and restore `gzip` compressed archives, and the `-j` option works with `bzipped` files and tarballs compressed with `xz`.

Creating Full and Incremental Backups with tar

If you want to create a full backup, the following creates a `bzip2` compressed tarball (the `j` option) of the entire system:

```
matthew@seymour:~$ sudo tar cjvf fullbackup.tar.bz2 /
```

To perform an incremental backup, you must locate all the files that have been changed since the last backup. For simplicity, assume that you do incremental backups on a daily basis. To locate the files, use the `find` command:

```
matthew@seymour:~$ sudo find / -newer name_of_last_backup_file ! -a -type f -print
```

When run alone, `find` generates a list of files system-wide and prints it to the screen. The `! -a -type` eliminates everything but regular files from the list; otherwise, the entire directory is sent to `tar`, even if the contents were not all changed.

Pipe the output of the `find` command to `tar` as follows:

```
matthew@seymour:~$ sudo find / -newer name_of_last_backup_file ! -type d -print |
\tar czT - backup_file_name_or_device_name
```

Here, the `T` - option gets the filenames from a buffer (where the `-` is the shorthand name for the buffer).

NOTE

The `tar` command can back up to a raw device (one with no file system) and to a formatted partition. For example, the following command backs up those directories to device `/dev/hdd` (not `/dev/hda1`, but to the unformatted device itself):

```
matthew@seymour:~$ sudo tar cvzf /dev/hdd /boot /etc /home
```

Restoring Files from an Archive with `tar`

The `xp` option with `tar` restores the files from a backup and preserves the file attributes, as well, and `tar` creates any subdirectories it needs. Be careful when using this option because the backups might have been created with either relative or absolute paths. You should use the `tvf` option with `tar` to list the files in the archive before extracting them so that you know where they will be placed.

For example, to restore a `tar` archive compressed with `bzip2`, use the following:

```
matthew@seymour:~$ sudo tar xjvf ubuntu-test.tar.bz2
```

To list the contents of a `tar` archive compressed with `bzip2`, use this:

```
matthew@seymour:~$ sudo tar tjvf ubuntu-test.tar.bz2
```

```
tar: Record size = 8 blocks
```

```
drwxr-xr-x matthew/matthew          0 2013-07-08 14:58 other/
```

```
-rwxr-xr-x matthew/matthew    1856 2013-04-29 14:37 other/matthew helmke
cccjpublic.asc
```

```
-rwxr-xr-x matthew/matthew      170 2013-05-28 18:11 backup.sh
-rwxr-xr-x matthew/matthew      1593 2013-10-11 10:38 backup method
```

Note that because the pathnames do not start with a backslash, they are relative pathnames and will install in your current working directory. If they were absolute pathnames, they would install exactly where the paths state.

The GNOME File Roller

The GNOME desktop file archiving graphical application File Roller (`file-roller`) views, extracts, and creates archive files using `tar`, `gzip`, `bzip`, `compress`, `zip`, `rar`, `lha`, and several other compression formats. Note that File Roller is only a front end to the command-line utilities that actually provide these compression formats; if a format is not installed, File Roller cannot use that format.

CAUTION

File Roller does not complain if you select a compression format that is not supported by installed software until after you attempt to create the archive. So be sure to install any needed compression utilities before you use File Roller.

File Roller is well integrated with the GNOME desktop environment to provide convenient drag-and-drop functionality with the Nautilus file manager. To create a new archive, select Archive, New to open the New Archive dialog box and navigate to the directory where you want the archive to be kept. Type your archive's name in the Selection: /root text box at the bottom of the New Archive dialog box. Use the Archive Type drop-down menu to select a compression method. Then drag the files that you want to be included from Nautilus into the empty space of the File Roller window, and the animated icons show that files are being included in the new archive. When you have finished, a list of files appears in the previously blank File Roller window. To save the archive, select Archive, Close. Opening an archive is as easy as using the Archive, Open dialog to select the appropriate archive file. You can learn more at <https://help.ubuntu.com/community/File%20Roller>.

The KDE `ark` Archiving Tool

Ubuntu also offers the KDE `ark` and `kdat` GUI tools for backups; they are installed only if you select the KDE desktop during installation, but you can search through Synaptic to find them. Archiving has traditionally been a function of system administrator and not seen as a task for individual users, so no elaborate GUI was believed necessary. Backing up has also been seen as a script-driven, automated task in which a GUI is not as useful. Although that's true for system administrators, home users usually want something a little more attractive and easier to use, and that's the gap `ark` was created to fill.

You launch `ark` by launching it from the command line. It is integrated with the KDE desktop (as File Roller is with GNOME), so it might be a better choice if you use KDE. This application provides a graphical interface for viewing, creating, adding to, and extracting from archived files. Several configuration options are available with `ark` to ensure its

compatibility with Microsoft Windows. You can drag and drop from the KDE desktop or Konqueror file browser to add or extract files, or you can use the `ark` menus.

As long as the associated command-line programs are installed, `ark` can work with `tar`, `gzip`, `bzip2`, `zip`, and `lha` files (the last four being compression methods used to save space through compaction of the archived files).

Existing archives are opened after launching the application itself. You can add files and directories to the archive or delete them from the archive. After opening the archive, you can extract all of its contents or individual files. You can also perform searches by using patterns (all `*.jpg` files, for example) to select files.

To create new archives, choose File, New, and then type the name of the archive, providing the appropriate extension (`.tar`, `.gz`, and so on). Then you can add files and directories as you desire.

Déjà Dup

Déjà Dup is a simple backup tool with a useful GUI. It supports local, remote, or cloud backups. It can encrypt and compress your data for secure and fast transfers and more. In the applications list, Ubuntu just calls it Backups (see Figure 17.1).



FIGURE 17.1 The Backups icon is easy to find.

After you open the Backups application, go through the menu items on the left to set where the backup will be stored, what will be backed up, a schedule for automatic backups, and more (see Figure 17.2). When you have set everything to your taste, remember to turn on Déjà Dup by toggling the setting at the upper right from Off to On.

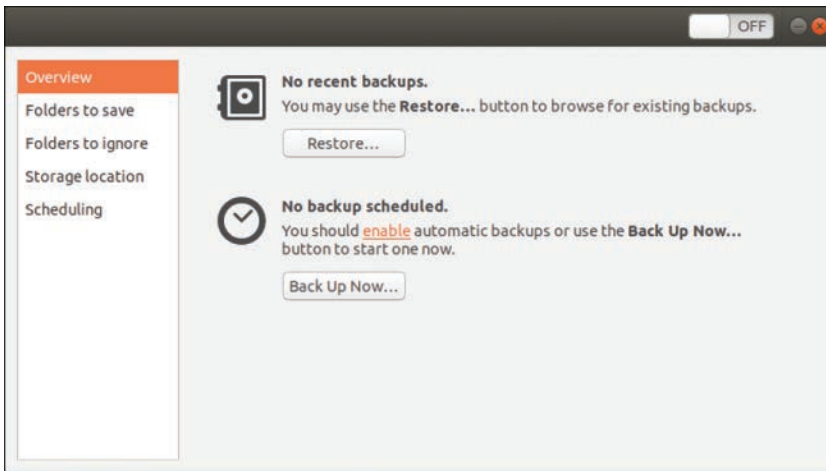


FIGURE 17.2 Backup settings are accessed using the menu entries on the left.

Back In Time

Back In Time is a viable alternative to Déjà Dup for many users. It is easily available from the Ubuntu software repositories, is stable, and has a clear and easy-to-understand interface.

Back In Time uses `rsync`, `diff`, and `cp` to monitor, create, and manipulate files, and it uses `cron` to schedule when it will run. Using these command-line tools is described later in this chapter. Back In Time is little more than a well-designed GUI front end designed for GNOME and also offers a separate package in the repositories with a front end for KDE. If you use the standard Ubuntu interface, install a package called `nautilus-actions` to get context menu access to some of the backup features.

The first time you run Back In Time, it takes a snapshot of your drive. This may take a long time, depending on the amount of data you have. You designate which files and directories to back up and where to back them up. Then set when to schedule the backup. The program takes care of the rest.

To restore, select the most recent snapshot from the list in Back In Time. Then browse through the list of directories and files until you find the file that interests you. You may right-click the file to view a pop-up menu, from which you may open a file, copy a file to a desired location, or view the various snapshots of a file and compare them to determine which one you might want to restore.

Back In Time keeps multiple logs of actions and activities, file changes, and versions, and it is a useful tool.

You can find the official documentation for Back In Time at <https://backintime.readthedocs.io/>.

Unison

Unison is a file-synchronization tool that works on multiple platforms, including Linux, other flavors of UNIX such as Solaris and macOS, and Windows. After Unison is set up, it synchronizes files in both directions and across platforms. If changes are made on both ends, files are updated in both directions. When file conflicts arise, such as when the same file was modified on each system, the user is prompted to decide what to do. Unison can connect across a network using many protocols, including `ssh`. It can connect with and synchronize many systems at the same time and even to the cloud.

Unison was developed at the University of Pennsylvania as a research project among several academics. It is no longer under active development as a research project, but it does appear to continue to be maintained with bug fixes and very occasional feature additions. The original developers claim to still be using it daily, so it is not completely abandoned.

Unison is powerful and configurable. The foundation is based on `rsync`, but with some additions that enable functionality that is generally available only from a version control system.

Even though the project is no longer the primary focus of any of the developers, many people still use Unison. For that reason, it gets a mention in this chapter and might be worthy of your time and effort if you are interested. Unison is released under the free GPL license, so you might decide you want to dig in to the code. The developers have publicly stated that they do not have time to maintain it regularly but welcome patches and contributions. If this is a project that interests you, see www.cis.upenn.edu/~bcpierce/unison/.

Amanda

Amanda is a powerful network backup application created by the University of Maryland at College Park. Amanda is a robust backup and restore application best suited to unattended backups with an autoloading tape drive of adequate capacity. It benefits from good user support and documentation.

Amanda's features include compression and encryption. It is intended for use with high-capacity tape drives, floptical, CD-R, and CD-RW devices.

Amanda uses GNU `tar` and `dump`; it is intended for unattended, automated tape backups and is not well suited for interactive or ad hoc backups. The support for tape devices in Amanda is robust, and file restoration is relatively simple. Although Amanda does not support older Macintosh clients, it uses Samba to back up Microsoft Windows clients, as well as any UNIX client that can use GNU tools (including macOS). Because Amanda runs on top of standard GNU tools, file restoration can be made using those tools on a recovery disk even if the Amanda server is not available. File compression can be done on either the client or server, thus lightening the computational load on less-powerful machines that need to be backed up.

CAUTION

Amanda does not support `dump` images larger than a single tape and requires a new tape for each run. If you forget to change a tape, Amanda continues to attempt backups until you insert a new tape, but those backups will not capture the data as you intended them to. Do not use a tape that is too small or forget to change a tape, or you will not be happy with the results.

There is no GUI for Amanda. Configuration is done in the time-honored UNIX tradition of editing text configuration files located in `/etc/amanda`. The default installation in Ubuntu includes a sample `cron` file because it is expected that you will be using `cron` to run Amanda regularly. The client utilities are installed with the package `amanda-client`; the Amanda server is called `amanda-server`. Install both. As far as backup schemes are concerned, Amanda calculates an optimal scheme on-the-fly and schedules it accordingly. It can be forced to adhere to a traditional scheme, but other tools are possibly better suited for that job.

The man page for Amanda (the client is `amdump`) is well written and useful, explaining both the configuration of Amanda and detailing the several programs that actually make up Amanda. The configuration files found in `/etc/amanda` are well commented; they provide a number of examples to assist you with configuration.

The program's home page is www.amanda.org. There you can find information about subscribing to the mail list and links to Amanda-related projects and a FAQ.

Alternative Backup Software

Commercial and other freeware backup products do exist; BRU and Veritas are good examples of effective commercial backup products. Here are some useful free software backup tools that are not installed with Ubuntu:

- ▶ **flexbackup**—This backup tool is a large file of Perl scripts that makes `dump` and `restore` easier to use. `flexbackup`'s command syntax can be found by using the command with the `-help` argument. It also can use `afio`, `cpio`, and `tar` to create and restore archives locally or over a network using `rsh` or `ssh` if security is a concern. Its home page is www.edwinh.org/flexbackup/. Note that it has not received any updates or changes in a very long time.
- ▶ **afio**—This tool creates `cpio` formatted archives but handles input data corruption better than `cpio` (which does not handle data input corruption very well at all). It supports multivolume archives during interactive operation and can make compressed archives. If you feel the need to use `cpio`, you might want to check out `afio`.

Many other alternative backup tools exist, but covering all of them is beyond the scope of this book.

Copying Files

Often, when you have only a few files that you need to protect from loss or corruption, it might make sense to simply copy the individual files to another storage medium rather than create an archive of them. You can use the `tar`, `cp`, `rsync`, and even `cpio` commands to do this; you can also use a handy file management tool known as `mc`. `tar` is the traditional choice because older versions of `cp` did not handle symbolic links and permissions well at times, causing those attributes (characteristics of the file) to be lost; `tar` handled those file attributes in a better manner. `cp` has been improved to fix those problems, but

`tar` is still more widely used. `rsync` is an excellent choice for mirroring sets of files, especially when done over a network.

To illustrate how to use file copying as a backup technique, the examples here show how to copy (not archive) a directory tree. This tree includes symbolic links and files that have special file permissions you need to keep intact.

Copying Files Using `tar`

One choice for copying files into another location is to use the `tar` command; you just create a `tar` file that is piped to `tar` to be uncompressed in the new location. To accomplish this, first change to the source directory. Then the entire command resembles this:

```
matthew@seymour:~$ tar -cvf files | (cd target_directory ; tar -xpf)
```

In this command, `files` is the filenames you want to include; you can use `*` to include the entire current directory.

When you change to the source directory and execute `tar`, you use the `cvf` arguments to do the following:

- ▶ `c`—Creates an archive.
- ▶ `v`—Specifies verbose; that is, lists the files processed so you can see that it is working.
- ▶ `f`—Specifies the filename of the archive. (In this case, it is `-`.)

The following `tar` command options can be useful for creating file copies for backup purposes:

- ▶ `l`—Stay in the local file system (so that you do not include remote volumes).
- ▶ `atime-preserve`—Do not change access times on files, even though you are accessing them now (to preserve the old access information for archival purposes).

The contents of the `tar` file (held for you temporarily in the buffer, which is named `-`) are then piped to the second expression, which extracts the files to the target directory. In shell programming (refer to Chapter 14), enclosing an expression in parentheses causes it to operate in a subshell and be executed first.

After you change to the target directory, you use the following options with `tar`:

- ▶ `x`—Extracts files from a `tar` archive.
- ▶ `p`—Preserves permissions.
- ▶ `f`—Specifies the filename, which in this case is `-`, the temporary buffer that holds the files archived with `tar`.

Compressing, Encrypting, and Sending tar Streams

The file copy techniques using the `tar` command in the previous section can also be used to quickly and securely copy a directory structure across a LAN or the Internet (using the `ssh` command). One way to make use of these techniques is to use the following command line to first compress the contents of a designated directory and then decompress the compressed and encrypted archive stream into a designated directory on a remote host:

```
matthew@seymour:~$ tar -cvzf data_folder | ssh remote_host '( cd ~/mybackup_dir;
tar -xvzf )'
```

The `tar` command is used to create, list, and compress the files in the directory named `data_folder`. The output is piped through the `ssh` (Secure Shell) command and sent to the remote computer named `remote_host`. On the remote computer, the stream is then extracted and saved in the directory named `/mybackup_dir`. You are prompted for a password to send the stream.

Copying Files Using cp

To copy files, you could use the `cp` command. The general format of the command when used for simple copying is as follows:

```
matthew@seymour:~$ cp -a source_directory target_directory
```

The `-a` argument is the same as `-dpr`:

- ▶ `-d`—Preserves symbolic links (by not dereferencing them) and copies the files that they point to instead of copying the links.
- ▶ `-p`—Preserves all file attributes, if possible. (File ownership might interfere.)
- ▶ `-R`—Copies directories recursively.

You can also use the `cp` command to quickly replicate directories and retain permissions by using it with the `-avR` command-line options. Using these options preserves file and directory permissions, gives verbose output, and recursively copies and re-creates subdirectories. You can also create a log of the backup during the backup by redirecting the standard output like this:

```
matthew@seymour:~$ sudo cp -avR directory_to_backup destination_vol_or_dir 1 > /
root/backup_log.txt
```

You can get the same effect this way:

```
matthew@seymour:~$ sudo cp -avR ubuntu /test2 1 > /root/backup_log.txt
```

This example makes an exact copy of the directory named `/ubuntu` on the volume named `/test2` and saves a backup report named `backup_log.txt` under `/root`.

Using `rsync`

An old favorite for backing up is `rsync`. One big reason for this is that `rsync` enables you to copy only files that have changed since the last backup. With this tool, although the initial backup might take a long time, subsequent backups are much faster. `rsync` is also highly configurable and can be used with removable media such as USB hard drives or over a network. Let's look at one way to use `rsync`.

First, create an empty file and call it `backup.sh`:

```
matthew@seymour:~$ sudo touch backup.sh
```

Then, using your favorite text editor, enter the following command into the file and save it:

```
rsync --force --ignore-errors --delete --delete-excluded --exclude-  
from=/home/matthew-exclude.txt --backup --backup-dir=`date +%Y-%m-%d` -av /  
/media/externaldrive/backup/Seymour
```

Make the file executable:

```
matthew@seymour:~$ sudo chmod +x backup.sh
```

This command uses several options with `rsync` and puts them in a script that is quick and easy to remember and run. You can run the script at the command line by using `sudo sh ./backup.sh` or as an automated `cron` job.

Here is a rundown of what is going on in the command. Basically, `rsync` is told to copy all new and changed files (what to back up) and delete from any existing backup any files that have been deleted on the source (and back them up in a special directory, just to be safe). It is told where to place the backup copy and is given details on how to deal with specific issues in the process. (Read the `rsync` man page for more options and to customize to your needs.)

Following are the options used here:

- ▶ `--force`—Forces deletion of directories in the target location that are deleted in the source, even if the directories in the destination are not empty.
- ▶ `--ignore-errors`—Tells `--delete` to go ahead and delete files even when there are I/O errors.
- ▶ `--delete`—Deletes extraneous files from destination directories.
- ▶ `--delete-excluded`—Also deletes excluded files from destination directories.
- ▶ `--exclude-from=/home/matt-exclude.txt`—Prevents backing up files or directories listed in this file. (It is a simple list with each excluded directory on its own line.)
- ▶ `--backup`—Creates backups of files before deleting them from a currently existing backup.

- ▶ `--backup-dir='date +%Y-%m-%d'`—Creates a backup directory for the previously mentioned files that looks like this: 2013-07-08. Why this format for the date? Because it is standard, as outlined in ISO 8601 (see www.iso.org/iso/home/standards/iso8601.htm). It is clear, works with scripts, and sorts beautifully, making your files easy to find.
- ▶ `-av`—Tells `rsync` to use archive mode and verbose mode.
- ▶ `/`—Denotes the directory to back up. In this case, it is the root directory of the source, so everything in the filesystem is being backed up. You could put `/home` here to back up all user directories or make a nice list of directories to exclude in the filesystem.
- ▶ `/media/externaldrive/backup/seymour`—Sets the destination for the backup as the `/backup/seymour` directory on an external hard drive mounted at `/mount/externaldrive`.

To restore from this backup to the same original location, you reverse some of the details and may omit others. Something like this works nicely:

```
matthew@seymour:~$ rsync --force --ignore-errors --delete --delete-excluded
/media/externaldrive/backup/seymour /
```

This becomes even more useful when you think of ways to script its use. You could create an entry in `crontab`, as described in Chapter 14. Even better, you could set two computers to allow for remote SSH connections using private keys created with `ssh-keygen`, as described in Chapter 19, so that one could back up the files from one computer to the other computer without requiring manual login. Then you could place that in an automated script.

Version Control for Configuration Files

For safety and ease of recovery when configuration files are corrupted or incorrectly edited, the use of a version control system is recommended. In fact, this is considered an industry best practice. Many top-quality version control systems are available, such as Git, Subversion, Mercurial, and Bazaar. If you already have a favorite, perhaps one that you use for code projects, you can do what we describe in this section using that version control system. The suggestions here are to get you thinking about the idea of using version control for configuration files and to introduce a few well-used and documented options for those who are unfamiliar with version control. First, some background.

Version control systems are designed to make it easy to revert changes made to a file, even after the file has been saved. Each system does this a little bit differently, but the basic idea is that not only is the current version of the file saved, but each and every version that existed previously is also saved. Some version control systems do this by saving the entire file every time. Some use metadata to describe just the differences between versions. In any case, it is possible to roll back to a previous version of the file to restore a file to a state before changes were made. Developers who write software are well aware of the

power and benefit to being able to do this quickly and easily; it is no longer required that the file editor remember the technical details of where, what, or even how a file has been edited. When a problem occurs, the file is simply restored to its previous state. The version control system is also able to inform the user where and how each file has changed at each save.

Using a version control system for configuration files means that every time a configuration is changed, those changes are recorded and tracked. This enables easy discovery of intruders (if a configuration has been changed by an unauthorized person trying to reset, say, the settings for Apache so that the intruder can allow a rogue web service or site to run on your server), easy recovery from errors and glitches, and easy discovery of new features or settings that have been enabled or included in the configuration by software upgrades.

Many older and well-known tools do this task, such as `changeltrack`, which is quite a good example. All such tools seek to make the job of tracking changes to configuration files more easily and quickly, but with the advances in version control systems, most provide very little extra benefit. Instead of suggesting any of these tools, we think you are probably better off learning a modern and good version control system. One exception is worth a bit of discussion because of its ability to work with your software package manager, which saves you the task of remembering to commit changes to your version control system each time the package manager runs. This exception is `etckeeper`.

`etckeeper` takes all of your `/etc` directory and stores the configuration files from it in a version control system repository. You can configure the program by editing the `etckeeper.conf` file to store data in a Git, Mercurial, Bazaar, or Subversion repository. In addition, `etckeeper` connects automatically to the APT package management tool used by Ubuntu and automatically commits changes made to `/etc` and the files in it during normal software package upgrades. Other package managers, such as Yum, can also be tracked when using other Linux distributions such as Fedora. It even tracks file metadata that is often not easily tracked by version control systems, like the permissions in `/etc/shadow`.

CAUTION

Using any version control system to track files that contain sensitive data such as passwords can be a security risk. Tracked files and the version control system itself should be treated with the same level of care as the sensitive data itself.

By default, `etckeeper` uses Git. On Ubuntu, this is changed to Bazaar (`bzr`) because it is the version control system used by Ubuntu developers. Because this is configurable, we mention just the steps here and leave it to you to adapt them for your particular favorite version control system.

First, edit `/etc/etckeeper/etckeeper.conf` to use your desired settings, such as the version control system to use, the system package manager being used, and whether to have changes automatically committed daily. After `etckeeper` is installed from the Ubuntu repositories, it must be initiated from the command line:

```
matthew@seymour:~$ etckeeper init
```


If you are only going to use `etckeeper` to track changes made to `/etc` when software updates are made using APT, you do not need to do anything else. If you edit files by hand, make sure you use your version control system's commands to commit those changes or use the following:

```
matthew@seymour:~$ etckeeper commit "Changed prompt style"
```

The message in quotes should reflect the change just made. This makes reading logs and finding exact changes much easier later.

Recovering or reverting file changes is then done using your version control system directly. Suppose, for example, that you have made a change in `/etc/bash.bashrc`, the file that sets the defaults for your `bash` shell. You read somewhere how to change the prompt and did not like the result. However, because the changes are being tracked, you can roll it back to the previous version. Because `bzr` is the default for `etckeeper` in Ubuntu, here is how you do that with `bzr`. First, check the log to find the commit number for the previous change:

```
matthew@seymour:~$ bzr log /etc/bash.bashrc
-----
revno: 2
committer: matthew <matthew@seymour>
branch nick: seymour etc repository
timestamp: Fri 2021-07-16 11:08:22 -0700
message:
    Changed /etc/bash.bashrc
-----
revno: 1
committer: matthew <matthew@seymour>
branch nick: seymour etc repository
timestamp: Fri 2021-07-16 11:00:16 -0700
message:
    Changed /etc/bash.bashrc
-----
```

If you know the change was made in the most recent revision, denoted `revno 2` (for revision number two), you can revert to that version:

```
matthew@seymour:~$ bzr revert -revision 2 /etc/bash.bashrc
```

Today it is common for programmers, systems administrators, and developer types to back up their dotfiles using version control. Dotfiles are the configuration files and directories in a user's `/home` directory, all of which begin with a dot, like `.bashrc`. These are not necessarily backed up by all software, and because they are often customized by highly technical people to suit their desires, backing them up is a good idea. Version control systems are commonly used. A program for Ubuntu called `dotdee` performs this task for a different type of configuration file or directory that ends with `.d` and is stored in `/etc`. You can find more information about `dotdee` in Chapter 9, "Managing Software."

System Rescue

There will come a time when you need to engage in system rescue efforts. This need arises when the system will not even start Linux so that you can recover any files. This problem is most frequently associated with the boot loader program or partition table, but it could be that critical system files have been inadvertently deleted or corrupted. If you have been making backups properly, these kinds of system failures are easily, though not quickly, recoverable through a full restore. Still, valuable current data might not have been backed up since the last scheduled backup, and the backup archives may be found to be corrupt, incomplete, or missing. A full restore also takes time you might not have. If the problem causing the system failure is simply a damaged boot loader, a damaged partition table, a missing library, or misconfiguration, a quick fix can get the system up and running, and the data can then be easily retrieved.

In this section, you learn a couple of quick things to try to restore a broken boot loader or recover your data when your system fails to boot.

The Ubuntu Rescue Disc

The Ubuntu installation DVD or USB drive works quite well as a live rescue system. To use it, insert the medium and reboot the computer, booting from it just as you did when you installed Ubuntu originally.

Restoring the GRUB2 Boot Loader

The easiest way to restore a broken system's GRUB2 files is simply to replace them. Your best bet is to use installation media from the same release as what you have installed on the hard drive.

To get started, boot using the live DVD and open a terminal. Then determine which of the hard drive's partitions holds the Ubuntu installation, which you can discover by using the following:

```
matthew@seymour:~$ sudo fdisk -l
```

You may find this block ID command useful, as it tends to return a bit more information:

```
matthew@seymour:~$ sudo blkid
```

Unless you customized your installation—in which case you probably already know your partitioning scheme and the location of your Ubuntu installation—the partition will probably be on a drive called `sda` on the first partition, which you can mount now by using this:

```
matthew@seymour:~$ sudo mount /dev/sda1 /mnt
```

This mounts the drive in the current file system (running from the live DVD) at `/mnt`, where it will be accessible to you for reading and modifying as needed. Next, you reinstall GRUB2 on this device:

```
matthew@seymour:~$ sudo grub-install --boot-directory=/mnt/boot /dev/sda
```

At this point, reboot (using your hard drive and not the live DVD), and all should be well. After the reboot is complete, enter the following:

```
matthew@seymour:~$ sudo update-grub
```

This refreshes the GRUB2 menu and completes the restoration. You can find a lot of great information about GRUB2 at <https://help.ubuntu.com/community/Grub2>.

Saving Files from a Nonbooting Hard Drive

If restoring the GRUB2 boot loader fails and you still cannot boot from the hard drive, try to use the live DVD to recover your data. Boot and mount the hard drive as shown previously and then attach an external storage device such as a USB thumb drive or an external hard drive. Then copy the files you want to save from the mounted drive to the external drive.

If you cannot mount the drive at all, your options become more limited and possibly more expensive. In this case, it is likely that either the hardware has failed or the file system has become badly corrupted. Either way, recovery is either impossible or more difficult and best left to experts if the data is important to you. But, the good news is that you have been making regular backups, right? So, you probably lost only a day or maybe a week of work and can buy a new drive, install it, and start from scratch, putting the data from your backup on your new Ubuntu installation on the new hardware.

Every experienced system administrator has had a drive fail; no hardware is infallible. We expect occasional hardware failures, and that's why we have good backup and recovery schemes in place for data. There are two types of system administrators: those who lose data when this happens and those who have good schemes in place. Be forewarned and be wise.

If you cannot boot a drive and do not have a backup, which happens to most system administrators only once in their lives (because they learn from the mistake), immediately stop messing with the hard drive. Your best bet to recover the data will be very expensive, but you should look for a company that specializes in the task and pay them to do it. If your data is not worth the expense for recovery and you want to try to recover it yourself, you can try, but this is not a task for the faint of heart, and more often than not, the data is simply lost. Again, the best course is to back up regularly, check your backups to be sure they are valid, and repeat. Practice restoring from backups before you need to do it, perhaps with a test system that is not vital and will not hurt anything if you make a mistake.

References

- ▶ <https://help.ubuntu.com/community/BackupYourSystem>—An excellent place to start for learning and examining backup methods in Ubuntu
- ▶ www.tldp.org—The Linux Documentation Project, which offers several useful HOWTO documents that discuss backups and disk recovery

CHAPTER 18

Networking

One of the benefits of open source technology in general and Linux in particular is that it can be used effortlessly across several networking environments and the Internet. With strong support for the standard Internet protocol TCP/IP, Linux can talk to all the UNIX flavors, including macOS, Windows (with the help of Samba), NetWare (IPX), and even older protocols such as DECnet and Banyan VINES. Many organizations use Linux as an Internet gateway, allowing many different clients to access the Internet through Linux, as well as communicate via email and instant messaging. Most important is its built-in support for IPv6, which has begun to see a significant uptake in the commercial/enterprise world. It's safe to say that whatever networking protocol you come across, Linux will be able to work with it in some way.

This chapter covers network and Internet connectivity, as most networks invariably end up connected to the Internet in some shape or form. You learn how to get the basics right, including configuration and management of *network interface cards* (NICs) and other network services with Ubuntu. You also find out how to manage network services from the command line—which is important in case you are ever confined to a command prompt. We also look at connectivity options, both for inbound and outbound network traffic, such as *Point-to-Point Protocol* (PPP).

We focus on the use of text interfaces and manual configurations in this chapter. We also include an overview of basic graphical network management in Ubuntu, which is becoming more and more popular. The *graphical user interface* (GUI) option has become much more stable, useful, and easy to comprehend, and most desktop users now use the GUI to interact with networking. However, this is a book

IN THIS CHAPTER

- ▶ Laying the Foundation: The `localhost` Interface
- ▶ Checking Connections with `ping`, `traceroute`, and `mtr`
- ▶ Networking with TCP/IP
- ▶ IPv6 Basics
- ▶ Network Organization
- ▶ Hardware Devices for Networking
- ▶ Using Network Configuration Tools
- ▶ Dynamic Host Configuration Protocol
- ▶ Wireless Networking
- ▶ Beyond the Network and onto the Internet
- ▶ Common Configuration Information
- ▶ References

for power users who want to learn about the guts of their system, so roll up your sleeves and prepare to get your hands dirty.

Laying the Foundation: The `localhost` Interface

The first thing that needs to be in place before you can successfully connect to the Internet or any other network is a `localhost` interface, sometimes called a *loopback interface*, and commonly referenced as `lo`. The TCP/IP protocol suite (see the section “Networking with TCP/IP,” later in this chapter) uses this interface to assign an IP address to your computer and is needed for Ubuntu to establish a PPP interface.

Checking for the Availability of the Loopback Interface

You should not normally have to manually create a loopback interface because Ubuntu creates one automatically for you during installation. To check that one is set up, you can use the `ip` command with a couple parameters to list all networking interfaces available, including the `lo` interface if it exists. This example shows only the information for `lo`:

```
matthew@seymour:~$ ip address show
1: lo: <LOOPBACK,UP,LOWER_UP> mtu 65536 qdisc noqueue state UNKNOWN group default qlen 1000
    link/loopback 00:00:00:00:00:00 brd 00:00:00:00:00:00
    inet 127.0.0.1/8 scope host lo
        valid_lft forever preferred_lft forever
    inet6 ::1/128 scope host
        valid_lft forever preferred_lft forever
```

What you see in this example is evidence that the loopback interface is present and active. The `inet` listed is the IP number assigned to the `localhost`, typically 127.0.0.1, along with the broadcast mask 255.0.0.0. You can also see the IPv6 address that is assigned to `lo`, which is `::1/128`, referred to as the `inet6`.

NOTE

Previously you checked for the availability of the loopback interface by using the now-deprecated `ifconfig`, like so:

```
matthew@seymour:~$ ifconfig
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:270 errors:0 dropped:0 overruns:0 frame:0
            TX packets:270 errors:0 dropped:0 overruns:0 carrier:0
            collisions:0 txqueuelen:0
            RX bytes:20748 (20.7 KB)  TX bytes:20748 (20.7 KB)
```

`ifconfig` still works but is slowly disappearing and is no longer installed by default. If you learned `ifconfig`, we strongly recommend spending the time to learn `ip` and use it instead.

Configuring the Loopback Interface Manually

The localhost interface's IP address is specified in a text configuration file that is used by Ubuntu to keep records of various network-wide IP addresses. The file is called `/etc/hosts` and usually exists on a system, even if it is empty. The Linux kernel and other networking tools use this file to access local IP addresses and hostnames. If you have not configured any other networking interfaces, you might find that the file looks something like this:

```
127.0.0.1    localhost
127.0.1.1    seymour

# The following lines are desirable for IPv6 capable hosts
::1        localhost ip6-localhost ip6-loopback
fe00::0    ip6-localnet
ff00::0    ip6-mcastprefix
ff02::1    ip6-allnodes
ff02::2    ip6-allrouters
```

The first line defines the special localhost interface and assigns it IP address 127.0.0.1. You might hear or read about terms such as localhost, loopback, and dummy interface; all these terms refer to the use of the IP address 127.0.0.1. The term loopback interface is used because, to Linux networking drivers, it looks as though the machine is talking to a network that consists of only one machine; the kernel sends network traffic to and from itself on the same computer. This is sometimes referred to as a dummy interface because the interface doesn't really exist; it is not a real address as far as the outside world is concerned; it exists only for the local machine, to trick the kernel into thinking that it and any network-aware programs running that require a network interface to operate have one available without them actually being aware that the connection is a connection to the same machine. It is a dummy not in the sense of stupid or silent, but in the sense that it is a mockup or substitute for something real.

Each networked Ubuntu machine on a LAN uses this same IP address for its localhost. If for some reason you discover that an Ubuntu computer does not have this interface, perhaps because some well-meaning person deleted it without understanding it was needed, you can use `sudo` and edit the `/etc/hosts` file to add the localhost entry as you saw previously and then use the `ifconfig` and `route` commands using your `sudo` permissions to create the interface, like this:

```
matthew@seymour:~$ sudo ip addr add 127.0.0.1/24 dev lo
matthew@seymour:~$ sudo ip route add 127.0.0.1/24 dev lo
```

These commands create the localhost interface in memory (all interfaces, such as `eth0` or `ppp0`, are created in memory when using Linux) and then add the IP address 127.0.0.1 to an internal (in-memory) table so that the Linux kernel's networking code can keep track of routes to different addresses.

Use the `ip` command as shown previously to test the interface.

Checking Connections with ping, traceroute, and mtr

If all worked properly in the preceding section, you should now be able to use the `ping` command to check that the interface is responding properly like this (using either `localhost` or its IP address):

```
matthew@seymour:~$ ping -c 3 localhost
PING localhost (127.0.0.1) 56(84) bytes of data.
64 bytes from localhost (127.0.0.1): icmp_seq=1 ttl=64 time=0.047 ms
64 bytes from localhost (127.0.0.1): icmp_seq=2 ttl=64 time=0.060 ms
64 bytes from localhost (127.0.0.1): icmp_seq=3 ttl=64 time=0.045 ms

--- localhost ping statistics ---
3 packets transmitted, 3 received, 0% packet loss, time 2030ms
rtt min/avg/max/mdev = 0.045/0.050/0.060/0.010 ms
```

You use the `-c` option to set the number of pings, and the command, if successful (as it was here), returns information regarding the round-trip speed of sending a test packet to the specified host.

The second line in the `/etc/hosts` file uses the actual hostname of the computer and assigns it to a similar private IP address that is unique to that computer. In the earlier code example, you can see that `127.0.1.1` is assigned to `seymour`, which is the name of the computer on which that `hosts` file resides.

The remaining lines are used for IPv6 and can be ignored—with the exception of the line that begins `::1`. This is used to define the `localhost` connection for IPv6, which you can test with the `ping6` command at the terminal, as follows:

```
matthew@seymour:~$ ping6 -c 3 ::1
PING ::1(::1) 56 data bytes
64 bytes from ::1: icmp_seq=1 ttl=64 time=0.072 ms
64 bytes from ::1: icmp_seq=2 ttl=64 time=0.065 ms
64 bytes from ::1: icmp_seq=3 ttl=64 time=0.061 ms
--- ::1 ping statistics ---
3 packets transmitted, 3 received, 0% packet loss, time 2060ms
rtt min/avg/max/mdev = 0.061/0.066/0.072/0.004 ms
```

This is a good place to pause and discuss three tools that are useful for checking a network: `ping/ping6`, `traceroute/traceroute6`, and `mtr`. A network timeout while you're using any of these tools indicates that a connectivity problem exists. If you get a response, then your network is working. Depending on the command, you might also receive information that helps you find and troubleshoot slow network problems.

You just used the first tool, `ping`, and its IPv6 version, `ping6`. These commands send a request to the specified network host (another computer that you specify on the same network), and if that computer receives the message, it sends a response. Using the `-c`

option followed by a number to limit the number of times the ping request is made is recommended. If that number is not stated, ping continues to make requests until you use Ctrl+C to stop the process. Here is an example, which is useful for determining whether your local connection is working:

```
matthew@seymour:~$ ping -c 3 google.com
PING google.com (74.125.225.103) 56(84) bytes of data.
64 bytes from ord08s08-in-f7.1e100.net (74.125.225.103): icmp_req=1 ttl=53 time=22.0 ms
64 bytes from ord08s08-in-f7.1e100.net (74.125.225.103): icmp_req=2 ttl=53 time=20.1 ms
64 bytes from ord08s08-in-f7.1e100.net (74.125.225.103): icmp_req=3 ttl=53 time=21.0 ms

--- google.com ping statistics ---
3 packets transmitted, 3 received, 0% packet loss, time 2004ms
rtt min/avg/max/mdev = 20.111/21.097/22.085/0.814 ms
```

The second tool, traceroute (along with its IPv6 version traceroute6), tracks the route that packets take on an IP network from the local computer to the network host specified. The traceroute6 tool is intended for use with IPv6, although it isn't necessary unless you want to force the command to trace using only IPv6—otherwise, traceroute tries to resolve the name given and automatically uses whichever protocol is most appropriate. Here is an example:

```
matthew@seymour:~$ traceroute google.com
traceroute to google.com (74.125.225.99), 30 hops max, 60 byte packets
 1 Cisco02420 (192.168.1.1)  0.149 ms  0.181 ms  0.304 ms
 2 10.2.0.1 (10.2.0.1)  3.190 ms  3.227 ms  3.217 ms
 3 65.201.51.216.sta.southslope.net (216.51.201.65)  3.397 ms  3.611 ms  3.720 ms
 4 ss-dsl-sec1.nl.southslope.net (167.142.151.30)  3.622 ms  3.637 ms  3.649 ms
 5 167.142.50.13 (167.142.50.13)  6.660 ms  6.665 ms  6.678 ms
 6 ins-dc2-et-8-4.desm.netins.net (167.142.67.17)  6.599 ms  6.503 ms  7.482 ms
 7 ins-db3-te-0-7-0-0.desm.netins.net (167.142.67.182)  7.845 ms  5.145 ms  5.131 ms
 8 216.176.4.29 (216.176.4.29)  20.557 ms  20.981 ms  20.978 ms
 9 216.176.4.58 (216.176.4.58)  20.124 ms  20.085 ms  20.103 ms
10 209.85.254.120 (209.85.254.120)  21.424 ms  22.390 ms  22.382 ms
11 209.85.240.150 (209.85.240.150)  23.318 ms  22.823 ms  22.821 ms
12 ord08s08-in-f3.1e100.net (74.125.225.99)  22.306 ms  23.269 ms  23.252 ms
```

The third tool, mtr, combines the functionality of ping and traceroute and gives you a live display of the data as it runs, as shown in this example:

```
My traceroute [v0.80]
example.lan Sat Jul 14 14:07:50 2012
```

Hostname	Packets				Pings		
	%Loss	Rcv	Snt	Last	Best	Avg	Worst
1. example.lan	0%	11	11	1	1	1	2
2. ae-31-51.ebr1.Chicago1.Level3.n	19%	9	11	3	1	7	14

3.	ae-1.ebr2.Chicago1.Level3.net	0%	11	11	7	1	7	14
4.	ae-2.ebr2.Washington1.Level3.ne	19%	9	11	19	18	23	31
5.	ae-1.ebr1.Washington1.Level3.ne	28%	8	11	22	18	24	30
6.	ge-3-0-0-53.gar1.Washington1.Le	0%	11	11	18	18	20	36
7.	63.210.29.230	0%	10	10	19	19	19	19
8.	t-3-1.bas1.re2.yahoo.com	0%	10	10	19	18	32	106
9.	p25.www.re2.yahoo.com	0%	10	10	19	18	19	19

`mtr` is not useful for creating a text file for analysis, but like the live system monitoring tool `top` (discussed in Chapter 16, “System-Monitoring Tools”), it gives real-time data and is quite powerful. As with `top`, you press the Q key to exit `mtr`.

Networking with TCP/IP

The basic building block for any network based on UNIX hosts is the *Transmission Control Protocol/Internet Protocol (TCP/IP)* suite, which includes three protocols even though only two appear in the name. The suite consists of *Internet Protocol (IP)*, *Transmission Control Protocol (TCP)*, and *User Datagram Protocol (UDP)*. The TCP/IP suite is *packet* based, which means that data is broken into little chunks on the transmit end for transmission to the receiving end. Breaking up data into manageable packets allows for faster and more accurate transfers. In TCP/IP, all data travels via IP packets, which is why addresses are referred to as *IP addresses*. IP is the lowest level of the suite.

TCP is also a connection-based protocol. Before data is transmitted between two machines, a connection is established between them. When a connection is made, a stream of data is sent to IP to be broken into the packets that are then transmitted. At the receiving end, the packets are put back in order and sent to the proper application port. TCP/IP forms the basis of the Internet; without it, the Internet would be a very different place indeed—if it even existed. In contrast, UDP is a connectionless protocol. Applications using this protocol just choose their destination and start sending. UDP is normally used for small amounts of data or on fast and reliable networks. If you are interested in the internals of TCP/IP, see the “References” section at the end of this chapter for places to look for more information.

UBUNTU AND NETWORKING

Chances are that your network card was configured during the installation of Ubuntu. You can use the `ip` command or Ubuntu’s graphical network configuration tools to edit your system’s network device information or to add or remove network devices on your system. Hundreds of networking commands and utilities are included with Ubuntu—far too many to cover in this chapter and more than enough to fill two or three volumes.

After reading this chapter, you might want to learn more about other graphical network clients for use with Linux. For example, you can use Nmap to scan a specific host for open ports and other running services (more at <https://nmap.org>). You may also find utilities like Netcat (more at <https://nc110.sourceforge.net>), Wireshark (more at www.wireshark.org), and `tcpdump` (more at www.tcpdump.org) useful.

TCP/IP Addressing

To understand networking with Linux, you need to know the basics of TCP/IP addressing. Internet IP addresses (also known as *public* IP addresses) are different from those used internally on a *local area network (LAN)*. Internet IP addresses are assigned (for the United States and some other hosts) by the American Registry for Internet Numbers (ARIN; see www.arin.net). Entities that need Internet addresses apply to this agency to be assigned addresses. ARIN assigns *Internet service providers (ISPs)* one or more blocks of IP addresses, which the ISPs can then assign to their subscribers.

You will quickly recognize the current form of TCP/IP addressing, known as *IP version 4 (IPv4)*. In this method, a TCP/IP address is expressed as a series of four decimal numbers: a 32-bit value expressed in a format known as dotted-decimal format, such as 192.168.0.1. Each set of numbers is known as an *octet* (eight 1s and 0s, such as 10000000 to represent 128) and ranges from 0 to 255.

The first octet usually determines what *class* the network belongs to. There are three classes of networks:

- ▶ **Class A**—Consists of networks with the first octet ranging from 1 to 126. There are only 126 Class A networks, each composed of up to 16,777,214 hosts. (If you are doing the math, there are potentially 16,777,216 addresses, but no host portion of an address can be all 0s or 255s.) The 10. network is reserved for local network use, and the 127. network is reserved for the loopback address, 127.0.0.1. TCP/IP uses loopback addressing to enable Linux network-related client and server programs to communicate on the same host. This address does not appear and is not accessible on your LAN.

NOTE

Notice that 0 is not included in Class A. The 0 address is used for network-to-network broadcasts. Also note that there are two other classes of networks, Classes D and E. Class D networks are reserved for multicast addresses and are not for use by network hosts. Class E addresses are deemed experimental and thus are not open for public addressing.

- ▶ **Class B**—Consists of networks defined by the first two octets, with the first ranging from 128 to 191. The 128. network is also reserved for local network use. There are 16,382 Class B networks, each with 65,534 possible hosts.
- ▶ **Class C**—Consists of a network defined by the first three octets with the first ranging from 192 to 223. The 192. network is another that is reserved for local network use. There are a possible 2,097,150 Class C networks of up to 254 hosts each.

No host portion of an IP address can be all 0s or 255s. These addresses are reserved for broadcast addresses. IP addresses with all 0s in the host portion are reserved for network-to-network broadcast addresses. IP addresses with all 255s in the host portion are reserved for local network broadcasts. Broadcast messages are not typically seen by users.

These classes are the standard, but a *netmask* also determines what class your network is in. The netmask determines what part of an IP address represents the network and what part represents the host. Common netmasks for the different classes are as follows:

- ▶ **Class A**—255.0.0.0
- ▶ **Class B**—255.255.0.0
- ▶ **Class C**—255.255.255.0

Because of the allocation of IP addresses for Internet hosts, getting a Class A network is now impossible. Getting a Class B network is also nearly impossible (all the addresses have been given out, but some companies are said to be willing to sell theirs), and Class C network availability is dropping rapidly, with the continued growth of Internet use worldwide.

LIMITS OF IPV4 ADDRESSING

The IPv4 address scheme is based on 32-bit numbering and limits the number of available IP addresses to about 4.1 billion. Many companies and organizations (particularly in the United States) were assigned very large blocks of IP addresses in the early stages of the growth of the Internet, which has left a shortage of “open” addresses. Even with careful allocation of Internet-connected host IP addresses and the use of *network address translation (NAT)* to provide communication to and from machines behind an Internet-connected computer, the Internet might run out of available addresses.

To solve this problem, a newer scheme named *IP version 6 (IPv6)* is being implemented. It uses a much larger addressing solution that is based on 128-bit addresses, with enough room to include much more information about a specific host or device, such as *Global Positioning Service (GPS)* or serial numbering. Although the specific details about the entire contents of the an IPv6 address have yet to be finalized, all Internet-related organizations appear to agree that something must be done to provide more addresses.

You can get a good overview of the differences between IPv4 and IPv6 policies regarding IP address assignments and the registration process of obtaining IP addresses at www.arin.net/knowledge/v4-v6.html and www.arin.net/resources/request.html.

Ubuntu, like all other modern Linux distributions, supports the use of IPv6 and includes a number of networking tools that conform to IPv6 addressing.

Migration to IPv6 is slow in coming, however, because many computer operating systems, software, hardware, firmware, and users are still in the IPv4 mindset. Supporting IPv6 requires rewriting many networking utilities, portions of operating systems currently in use, and firmware in routing and firewall hardware.

See the “IPv6 Basics” section, later in this chapter, for more on IPv6.

Using IP Masquerading in Ubuntu

Three blocks of IP addresses are reserved for use on internal networks and hosts not directly connected to the Internet. The address ranges are from 10.0.0.0 to 10.255.255.255, or 1 Class A network; from 172.16.0.0 to 172.31.255.255, or 16 Class B networks; and

from 192.168.0.0 to 192.168.255.255, or 256 Class C networks. Use these IP addresses when building a LAN for your business or home. Which class you choose can depend on the number of hosts on your network.

Internet access for your internal network can be provided by another PC or a router. The host or device is connected to the Internet and is used as an Internet gateway to forward information to and from your LAN. The host should also be used as a firewall to protect your network from malicious data and users while functioning as an Internet gateway.

A PC used in this fashion typically has at least two network interfaces. One is connected to the Internet and the other is connected to the computers on the LAN (via a hub or switch). Some broadband devices also incorporate four or more switching network interfaces. Data is then passed between the LAN and the Internet via NAT, sometimes known in networking circles as *IP masquerading*.

NOTE

Do not rely on a single point of protection for your LAN, especially if you use wireless networking, provide dial-in services, or allow mobile (laptop or PDA) users internal or external access to your network. Companies, institutions, and individuals that rely on a “moat mentality” have often discovered to their dismay that such an approach to security is easily breached. Make sure that your network operation is accompanied by a security policy that stresses multiple levels of secure access, with protection built into every server and workstation—something easily accomplished when using Linux.

Ports

Most servers on your network perform more than one task. For example, web servers often have to serve both standard and secure pages. You might also be running an FTP server on the same host. For this reason, applications are provided *ports* to use to make “direct” connections for specific software services. These ports help TCP/IP distinguish services so that data can get to the correct application. If you check the file `/etc/services`, you see the common ports and their usage. For example, for FTP, HTTP, and POP3 (email retrieval server), you see the following:

```
ftp      21/tcp
http     80/tcp      http      # WorldWideWeb HTTP
pop3     110/tcp     pop-3     # POP version 3
```

The ports defined in `/etc/services` in this example are 21 for FTP, 80 for HTTP, and 110 for POP3. Some other common port assignments are 25 for *Simple Mail Transfer Protocol* (SMTP) and 22 for *Secure Shell* (SSH) remote login. Note that these ports are not set in stone, and you can set up your server to respond to different ports. For example, although port 22 is listed in `/etc/services` as a common default for SSH, you can configure the `sshd` server to listen on a different port by editing its configuration file, `/etc/ssh/sshd_config`. The default setting (commented out with a pound sign, #) looks like this:

```
#Port 22
```

Edit the entry to use a different port, making sure to select an unused port number, as follows:

```
Port 2224
```

Save your changes and then restart the `sshd` server with `sudo service ssh restart`. Remote users must now access the host through port 2224, which can be done using `ssh's` `-p` (port) option, like this:

```
matthew@seymour:~$ ssh -p 2224 remote_host_name_or_IP
```

IPv6 Basics

Much of what this chapter discusses is valid regardless of whether you are using IPv4 or IPv6. We start here with a short description of each to lay a foundation for further understanding. As IPv6 receives greater acceptance and use, this understanding should be adequate to help you transition between the two, even if specific issues are not addressed in the chapter. If you missed the “Limits of IPv4 Addressing” note in the earlier “TCP/IP Addressing” section, you should go back and read through it to get started.

IPv4 is based on 32-bit numbering, which limits the number of available IP addresses to about 4.1 billion. This, and how those addresses were assigned, have led to the realization that not enough IPv4 addresses are available for the number of devices that need IP addresses. This problem, noticed in the 1990s, is only one of the problems with IPv4. Others include large routing tables, which are lists of the routes to particular network destinations, and sometimes the network distances and topography associated with those routes. These tables are stored in routers and networked computers.

To deal with these issues, IPv6 uses 128-bit numbering that can theoretically allow well over 340,282,366,920,938,463,463,374,607,431,768,211,456 IP addresses, which is normally expressed in scientific notation as about 3.4×10^{38} addresses. That's about 340 trillion, trillion, trillion addresses, meaning we are unlikely to run out again anytime soon. This number of addresses allows for each computer to have its own globally routable address. We don't need NAT in IPv6 to translate IP addresses as packets pass through a routing device because an adequate number of addresses are available. IPv6 allows us to go back to the easier-to-configure peer-to-peer style of Internet networking originally conceived of and used in the 1980s. This creates routing tables that are much smaller because fewer subroutes need to be generated.

Some other useful features of IPv6 include the following:

- ▶ Address autoconfiguration (RFC2462)
- ▶ Anycast addresses (“one-out-of-many”)
- ▶ Mandatory multicast addresses
- ▶ IPsec (IP Security)
- ▶ Simplified header structure

- ▶ Mobile IP
- ▶ IPv6-to-IPv4 transition mechanisms

There are different types of IPv6 addresses. Unicast addresses are the well-known addresses; packets sent to these addresses arrive directly at the interface that belongs to the address. Anycast addresses look the same as unicast addresses, but they actually address a group of interfaces; packets sent to an anycast address arrive at the nearest (in the router metric sense) interface. Anycast addresses may only be used by routers. Finally, a multicast address identifies a group of interfaces; packets sent to a multicast address arrive at all interfaces belonging to the multicast group.

IPv6 addresses are created using eight sets of numbers, like this:

```
F734:0000:0000:0000:3458:79B2:D07B:4620
```

Each of the eight sections is made of a four-digit number in hexadecimal, which means that each digit can be from 0 to 9 or A to F (A=10, B=11, and so on). Hexadecimal is a denser format than binary. In binary, there are only two options, 0 and 1. This means that in hexadecimal, 4 digits can be used to represent 16 binary digits, like this:

- ▶ Binary 0000000000000000 = hex 0000 (or just 0)
- ▶ Binary 1111111111111111 = hex FFFF
- ▶ Binary 1101010011011011 = hex D4DB

So, a 128-bit address written in binary would be very long indeed. This 128-bit address written in binary and separated by dots:

```
1111111111111111.1111111111111111.1111111111111111.1111111111111111.1111111111111111.1111111111111111.1111111111111111.1111111111111111
```

is the same as this 128-bit address, written in hexadecimal and separated by colons:

```
FFFF:FFFF:FFFF:FFFF:FFFF:FFFF:FFFF:FFFF
```

So, understandably, we use the latter hexadecimal format for IPv6 (and the binary format is not used, just in case you were wondering).

Often an address has long substrings of all zeros; the longest and first run of all-zero sections is abbreviated as a double colon (::). Because :: is variable in length, it can be used only once per address. Leading 0s—up to three per section—are also omitted. When this is done, the result is called the *canonical* form. For example, `fe80::1` is the canonical form of `fe80:0000:0000:0000:0000:0000:0000:0001`, and `2001:db8:b:23c1:49:4592:efe:9982` is the canonical form of `2001:0db8:000b:23c1:0049:4592:0efe:9982`.

Writing the last 32 bits of an IPv6 address using the well-known IPv4 format is also possible. For example, `2002::10.0.0.1` corresponds to the long form `2002:0000:0000:0000:0000:0000:0a00:0001`, which then can be compressed to the canonical form `2002::a00:1`.

As in IPv4, an IPv6 address has sections for the network and for the device. However, an IPv6 address has a dedicated section for subnetting. The following examples use 1s to show the section of the address being described (in binary because that is easier for us humans) and 0s for the rest of the address.

In IPv6, the first 48 bits are for Internet routing (network addressing):

```
11111111111111111111.11111111111111111111.11111111111111111111.0000000000000000. 0000000000000000.0000000000000000.0000000000000000.0000000000000000
```

The 16 bits from the 49th to the 54th are for defining subnets:

```
0000000000000000.0000000000000000.0000000000000000.1111111111111111. 0000000000000000.0000000000000000.0000000000000000.0000000000000000
```

The last 64 bits are for device (interface) IDs:

```
0000000000000000.0000000000000000.0000000000000000.0000000000000000. 1111111111111111.1111111111111111.1111111111111111.1111111111111111
```

It is easier for humans to conceive of these using binary, but to actually use this information, you have to convert numbers from binary to hexadecimal. Fortunately, this is easily accomplished on the Web using a quick Google search for “binary to hex” conversion.

Let’s say you want to break your corporate network into 64 subnets. The binary mask just for the subnetting range would be 1111110000000000, which translates to a hex value of FC00. Some IPv6 masking tools work with just this one hex word; otherwise, a full 128-bit hex mask would be FFFF:FFFF:FFFF:FC00:0:0:0:0.

Here are some special-use, reserved IPv6 addresses:

- ▶ ::1/128 is the loopback address.
- ▶ ::/128 is the unspecified address.
- ▶ ::IPv4-address/96 are the IPv4-compatible addresses.
- ▶ The 2001:db8::/32 are the documentation addresses. They are used for documentation purposes such as user manuals, RFCs, and so on.
- ▶ ::/0 is the default unicast route address.
- ▶ ff00::/8 are multicast addresses.

This section of the book is certain to grow as time passes and IPv6 becomes more commonly used. For now, this introduction is probably all you are likely to need, especially since IPv4 is not going away. This transition is a process of adding IPv6 into existing worldwide networking schemes and system abilities and is neither intended nor likely to completely replace IPv4.

Network Organization

Properly organizing your network addressing process grows more difficult as the size of your network grows. Setting up network addressing for a Class C network with fewer than 254 devices is simple. Setting up addressing for a large, worldwide company with a Class A network and many different users can be extremely complex. If your company has fewer than 254 *hosts* (meaning any device that requires an IP address, including computers, printers, routers, switches, and other devices) and all your workgroups can share information, a single Class C network is sufficient.

Subnetting

Within Class A and B networks, there can be separate networks called *subnets*. Subnets are considered part of the host portion of an address for network class definitions. For example, in the 128. Class B network, you can have one computer with address 128.10.10.10 and another with address 128.10.200.20; these computers are on the same network (128.10.), but they have different subnets (128.10.10. and 128.10.200.). Because of this, communication between the two computers requires either a router or a switch. Subnets can be helpful for separating workgroups within a company.

Often subnets can be used to separate workgroups that have no real need to interact with or to shield from other groups' information passing among members of a specific workgroup. For example, if your company is large enough to have its own HR department and payroll section, you could put those departments' hosts on their own subnet and use your router configuration to limit the hosts that can connect to this subnet. This configuration prevents networked workers who are not members of the designated departments from being able to view some of the confidential information the HR and payroll personnel work with.

Subnet use also enables your network to grow beyond 254 hosts and share IP addresses. With proper routing configuration, users might not even know they are on a different subnet from their co-workers. Another common use for subnetting is with networks that cover a wide geographic area. It is not practical for a company with offices in Chicago and London to have both offices on the same subnet, so using a separate subnet for each office is the best solution.

Subnet Masks

TCP/IP uses subnet masks to show which part of an IP address is the network portion and which part is the host. Subnet masks are usually referred to as *netmasks*. For a pure Class A network, the netmask is 255.0.0.0; for a Class B network, the netmask is 255.255.0.0; and for a Class C network, the netmask is 255.255.255.0. You can also use netmasks to deviate from the standard classes.

By using customized netmasks, you can subnet your network to fit your needs. For example, say that your network has a single Class C address. You have a need to subnet your network. Although this is not possible with a normal Class C subnet mask, you can

change the mask to break your network into subnets. By changing the last octet to a number greater than zero, you can break the network into as many subnets as you need.

For more information on how to create customized subnet masks, see Day 6, “The Art of Subnet Masking,” in *Sams Teach Yourself TCP/IP Network Administration in 21 Days*. That chapter goes into great detail on how to create custom netmasks and explains how to create an addressing cheat sheet for hosts on each subnet. The Linux Network Administrators Guide, at www.tldp.org/LDP/nag2/index.html, also has good information about how to create subnets.

Broadcast, Unicast, and Multicast Addressing

Information can get to systems through three types of addresses: unicast, multicast, and broadcast. Each type of address is used according to the purpose of the information being sent, as explained here:

- ▶ **Unicast**—Sends information to one specific host. Unicast addresses are used for Telnet, FTP, SSH, or any other information that needs to be shared in a one-to-one exchange of information. Although it is possible that any host on the subnet/network can see the information being passed, only one host is the intended recipient and will take action on the information being received.
- ▶ **Multicasting**—Broadcasts information to groups of computers sharing an application, such as a video conferencing client or an online gaming application. All the machines participating in the conference or game require the same information at precisely the same time to be effective.
- ▶ **Broadcasting**—Transmits information to all the hosts on a network or subnet. *Dynamic Host Configuration Protocol (DHCP)* uses broadcast messages when the DHCP client looks for a DHCP server to get its network settings, and *Reverse Address Resolution Protocol (RARP)* uses broadcast messages for hardware address-to-IP address resolution. Broadcast messages use .255 in all the host octets of the network IP address. (10.2.255.255 broadcasts to every host in your Class B network.)

Hardware Devices for Networking

As stated at the beginning of this chapter, networking is one of the strong points of the Linux operating system. This section covers the classes of devices used for basic networking. Note that this section talks about hardware devices, and not Linux networking devices, which are discussed in the section “Using Network Configuration Tools.”

Network Interface Cards

A computer must have a *network interface card (NIC)* to connect to a network. Currently, there are several topologies (ways of connecting computers) for network connections. These topologies range from the old and mostly outdated 10BASE-2 to the much newer and popular wireless Wi-Fi, or 802.11, networking.

Each NIC has a unique address (the hardware address, known as *Media Access Control [MAC]* address), which identifies that NIC. This address is six pairs of hexadecimal bits separated

by colons (:). A MAC address looks similar to this: 00:60:08:8F:5A:D9. The hardware address is used by DHCP (see the section “Dynamic Host Configuration Protocol,” later in this chapter) to identify a specific host. In addition, *Address Resolution Protocol (ARP)* and *Reverse Address Resolution Protocol (RARP)* use MAC addresses to map hosts to IP addresses.

This section covers some of the different types of NICs used to connect to a network, *including several that have long been obsolete* but that you may still find in use in older systems.

Token Ring

Token Ring networking was developed by IBM. As the name implies, the network is set up in a ring. A single “token” is passed from host to host, indicating the receiving host’s permission to transmit data.

Token Ring has a maximum transfer rate of 16Mbps (16 million bits per second). Unlike 10BASE-2 and 10BASE-5, Token Ring uses what is called *unshielded twisted pair (UTP)* cable. This cable looks a lot like the cable that connects your phone to the wall. Almost all Token Ring NICs are recognized by Linux.

10BASE-T

10BASE-T was the standard for a long time. A large number of networks still use it. Like Token Ring, 10BASE-T also uses UTP cable. Instead of being configured in a ring, 10BASE-T mostly uses a star architecture. In this architecture, the hosts all connect to a central location (usually a hub, which you learn about later in this chapter, in the “Hubs and Switches” section). All the data is sent to all hosts, but only the destination host takes action on individual packets. 10BASE-T has a transfer rate of 10Mbps.

10BASE-T has a maximum segment length of 100 meters (about 325 feet). There are many manufacturers of 10BASE-T NICs, and most are recognized by Ubuntu.

100BASE-T

100BASE-T was popular around the turn of the millennium, offering the same ease of administration as 10BASE-T while increasing the speed by a factor of 10. For most networks, the step from 10BASE-T to 100BASE-T is as simple as replacing NICs and hubs. Most 100BASE-T NICs and hubs can also handle 10BASE-T and can automatically detect which is in use. This allows for gradual network upgrades and usually does not require rewiring the whole network. Nearly every known 100BASE-T NIC and most generic NICs are compatible with Linux. 100BASE-T requires Category 5 UTP cabling.

1000BASE-T

1000BASE-T—usually referred to as *Gigabit Ethernet*—is the long-time standard in enterprise networking. Like 100BASE-T NICs, gigabit NICs automatically downgrade if they are plugged in to a slower network. Also, as with 100BASE-T, gigabit NICs require Category 5 UTP cabling; however, many institutions are now deploying Category 6 cables, which have much longer range and so are often worth the extra cost. You will find that most newer computers are sold with gigabit NICs. Fiber-related gigabit that uses fiber optics is termed *1000BASE-X*, whereas 1000BASE-T Gigabit Ethernet uses twisted-pair cabling (see the “Unshielded Twisted Pair” section, later in this chapter).

10G Ethernet and 50G Ethernet

10G Ethernet is now the most commonly used standard and has replaced 1000BASE-T in the majority of datacenters. It transmits at 10 gigabits per second, which is 10 times faster. 10G can use either copper or fiber cabling. If you use copper, you must use higher-grade cables if you want to run distances up to the stated 100 meter lengths. Fiber is really the norm here for all but short lengths.

50G Ethernet is the up-and-coming standard. It transmits at 50 gigabits per second and should be the choice when starting fresh with a new datacenter. Existing datacenters seem to be switching over gradually as they perform scheduled system decommissioning and replacements.

Fiber-Optic

Fiber-optic is more commonly used in newer and high-end installations because the cost of upgrading can be prohibitive for older sites.

Fiber-optic cable was originally used on *fiber distributed data interface (FDDI)* networks, similar to Token Ring in structure except that there are two rings (one primary, the other secondary). The primary ring is used exclusively, and the secondary sits idle until there is a break in the primary ring. That is when the secondary ring takes over, keeping the network alive. FDDI has a speed of 100Mbps and has a maximum ring length of 100 kilometers (62 miles). FDDI uses several tokens at the same time that, along with the faster speed of fiber-optics, account for the drastic increase in network speed.

As stated earlier, switching to a fiber-optic network can be very costly. To make the upgrade, the whole network has to be rewired, and all NICs must be replaced at the same time. Most FDDI NICs are recognized by Linux.

Wireless Network Interfaces

Wireless networking, as the name states, works without network cables; it is an extremely popular option. Upgrading is as easy as replacing network cards and equipment, such as routers and switches. Wireless networking equipment can also work along with the traditional wired networking to continue using existing equipment.

A wireless network is still generally slower than a traditional wired network. However, this situation is changing with wider adoption of newer protocols.

Network Cable

Currently, three types of network cable are available: coaxial, UTP, and fiber. Coaxial cable looks a lot like the coaxial cable used to connect your television to the cable jack or antenna. UTP looks a lot like the cable that runs from your phone to the wall jack (though the jacks are a bit wider). Fiber cable looks sort of like the RCA cables used on a stereo or like the cable used on electrical appliances in a home (with two separate segments connected together). The following sections discuss UTP and fiber network cable in more detail.

Unshielded Twisted Pair

UTP uses color-coded pairs of thin copper wire to transmit data. Each of the six categories of UTP serves a different purpose:

- ▶ **Category 1 (Cat1)**—Used for voice transmissions such as phone. Only one pair is used per line: one wire to transmit and one to receive. An RJ-11 plug is used to connect the cable from the phone to the wall.
- ▶ **Category 2 (Cat2)**—Used in early Token Ring networks. Has a transmission rate of 4Mbps and has the slowest data transfer rate. An RJ-11 plug is used for cable connections.
- ▶ **Category 3 (Cat3)**—Used for 10BASE-T networks. It has a transmission rate of 10Mbps. Three pairs of cables are used to send and receive signals. RJ-11 or RJ-45 plugs can be used for Cat3 cables, usually deferring to the smaller RJ-11. RJ-45 plugs are similar in design to RJ-11 but are larger to handle up to four pairs of wire and are used more commonly on Cat5 cables.
- ▶ **Category 4 (Cat4)**—Used in modern Token Ring networks. It has a transmission rate of 16Mbps and is becoming less and less common as companies are switching to better alternatives. RJ-45 plugs are used for cable connections.
- ▶ **Category 5 (Cat5)**—The fastest of the UTP categories, with a transmission rate up to 1000Mbps. It is used in both 100BASE-T and 1000BASE-T networks and uses four pairs of wire. Cat5 cable came out just as 10BASE-T networks were becoming popular, and it isn't much more expensive than Cat3 cable. As a result, most 10BASE-T networks use Cat5 UTP rather than Cat3. Cat5 cable uses RJ-45 plugs. Cat 5e (which stands for Category 5 enhanced) cable is similar to basic Cat 5, except that it fulfills higher standards of data transmission. While Cat 5 is common in existing cabling systems, Category 5e has almost entirely replaced it in new installations. Cat 5e can handle data transfer at 1000Mbps, is suitable for Gigabit Ethernet, and experiences much lower levels of near-end crosstalk (NEXT) than Cat 5.
- ▶ **Category 6 (Cat6)**—Also rated at 1000Mbps, this cable is available in two forms: stranded for short runs (25-meter runs, about 80 feet) and solid for up to 100-meter runs (about 325 feet), though the solid form should not be flexed.
- ▶ **Category 7 (Cat7)**—Rated for a transmission speed of up to 10Gbps and backward compatible with Cat6, Cat5, and Cat5e, Cat7 provides a 100 meter four-conductor channel. It requires twisted wires for full shielding and eliminates crosstalk with better noise resistance. The main objective is to get higher speeds with longer cables.
- ▶ **Category 8 (Cat8)**—Cat8 is a new type of cable supporting speeds up to 40Gbps. It is limited to a 30 meter two-conductor channel. The trade-off here requires asking whether you need the increased speed or rather the increased cable length in Cat7.
- ▶ **Category 9 (Cat9)**—At the time of this writing, Cat9 is being discussed but the standard is not finalized. It may not matter. Cat 6, 7, and 8 are easily good enough for today's needs. By the time most installations are going to consider Cat9, they are probably moving to fiber.

Fiber-Optic Cable

Fiber-optic cable (fiber) is usually orange or red in color. The transmission rate is 100Mbps over a maximum length of 100 kilometers (62 miles) or faster over short distances, in the range of 100Gbps in distances under 100 meters. Fiber uses a two-pronged plug to connect to devices. Fiber provides a couple of advantages because it uses light rather than electricity to transmit signals: It is immune to electromagnetic interference, and it is also more difficult to tap into and eavesdrop.

Hubs and Switches

Hubs and switches are used to connect several hosts together in a star architecture network. They can have any number of connections; the common sizes are 4, 8, 16, 24, and 48 connections (ports), and each port has a light that comes on when a network connection is made (link light). Hubs and switches enable you to expand your network easily; you can just add new hubs or switches when you need to add new connections. Each unit can connect to the other hubs or switches on the network, typically through a port on the hub or switch called an *uplink* port. This means two hubs or switches, connected by their uplink ports, can act as one hub or switch. Having a central location where all the hosts on your network can connect allows for easier troubleshooting of problems. If one host goes down, none of the other hosts are affected (depending on the purpose of the downed host). Because hubs and switches are not directly involved with the Linux operating system, compatibility is not an issue.

If you are constructing a small to midsize network, it is important to consider whether you intend to use either hubs or switches. Hubs and switches are visually the same in that they have rows of network ports. However, under the hood, the difference is quite important. Data is sent as packets of information across the network; with a hub, the data is transmitted simultaneously to all the network ports, regardless of which port the destination computer is attached to.

Switches, however, are more intelligent because they can direct packets of information to the correct network port that leads to the destination computer. They do this by “learning” the MAC addresses of each computer that is attached to them. In short, using switches minimizes excess packets being sent across the network, thus increasing network bandwidth available. In a small network with a handful of computers, the use of hubs might be perfectly acceptable, and you will find that hubs are generally cheaper than switches. However, for larger networks of 15 computers or more, you should consider implementing a switched network.

TIP

Troubleshooting network connections can be challenging, especially on large networks. If a user complains that he has lost his network connection, examining the hub or switch is a good place to start. If the link light for the user’s port is lit, chances are the problem is with the user’s network configuration. If the link light is not on, the host’s NIC is bad, the cable is not inserted properly, or the cable has gone bad for some reason.

Routers and Bridges

Routers and bridges are used to connect different networks to your network and to connect different subnets within your network. Routers and bridges both serve the same purpose of connecting networks and subnets, but they do so using different techniques. The information in the following sections helps you choose the connection method that best suits your needs.

Bridges

Bridges are used within a network to connect different subnets. A bridge blindly relays all information from one subnet to another without any filtering and is often referred to as a *dumb gateway*. This can be helpful if one subnet in your network is becoming overburdened and you need to lighten the load. A bridge is not very good for connecting to the Internet, however, because it lacks filtering. You really do not want all traffic traveling the Internet to be able to get through to your network.

Routers

Routers can pass data from one network to another, and they allow for filtering of data. Routers are best suited to connect your network to an outside network, such as the Internet. If you have a web server for an internal intranet that you do not want people to access from the Internet, for example, you can use a router's filter to block port 80 from outside your internal network. These filters can be used to block specific hosts from accessing the Internet, as well. For these reasons, routers are also called *smart gateways*.

Routers range in complexity and price from an enterprise-grade Cisco brand router that can cost thousands of dollars to consumer brands designed for home or small office use that can cost less than \$50.

Initializing New Network Hardware

All the initial network configuration and hardware initialization for Ubuntu is normally done during installation. At times, however, you may have to reconfigure networking on your system, such as when a host needs to be moved to a different subnet or a different network, or if you replace any of your computer's networking hardware.

Linux creates network interfaces in memory when the kernel recognizes that a NIC or another network device is attached to the system. These interfaces are unlike other Linux interfaces, such as serial communications ports, and they do not have a corresponding device file in the `/dev` directory. Unless support for a particular NIC is built in to your kernel, Linux must be told to load a specific kernel module to support your NIC. More than 100 such modules are located in the `/lib/modules/5.5.XX-XX/kernel/net` directory (where `XX-XX` is your version of the kernel).

You can initialize a NIC in several ways when using Linux. When you first install Ubuntu, automatic hardware probing detects and configures your system to use any installed NICs. If you remove the original NIC and replace it with a different make and model, your system will not automatically detect and initialize the device unless you configure Ubuntu to use automatic hardware detection when booting. Ubuntu should detect the absence of the old NIC and the presence of the new NIC at boot time.

If you do not use automatic hardware detection and configuration, you can initialize network hardware by doing the following:

- ▶ Manually editing the `/etc/modprobe.conf` file to prompt the system to recognize and support the new hardware upon reboot
- ▶ Manually loading or unloading the new device's kernel module with the `modprobe` command

The following sections explain these methods in greater detail.

Editing the `/etc/modprobe.conf` File

The `/etc/modprobe.conf` file might not be present when you first look for it, so you might need to create a blank file in a text editor. You can manually edit the `/etc/modprobe.conf` file to add a module dependency entry (also known as a *directive*) to support a new NIC or another network device. This entry includes the device's name and its corresponding kernel module. After you add this entry and reboot, the Linux kernel recognizes your new networking hardware. Ubuntu runs a module dependency check upon booting.

For example, if your system uses a RealTek NIC, you could use an entry like this:

```
alias eth0 8139too
```

This entry tells the Linux kernel to load the `8139too.o` kernel module to support the `eth0` network device.

On the other hand, if you have an Intel Ethernet Pro NIC installed, you use an entry like this:

```
alias eth0 eepro100
```

You can pass other parameters to a kernel module using one or more optional entries, if needed, to properly configure your NIC. See the `modprobe.conf` man page for more information about using entries. For more specifics regarding NIC kernel modules, examine the module's source code. (No man pages are yet available—which presents a good opportunity for anyone willing to write the documentation.)

In addition, check the `/etc/modprobe.d` directory for other files related to kernel modules.

Using `modprobe` to Manually Load Kernel Modules

You do not have to use an `/etc/modprobe.conf` entry to initialize kernel support for your new network device. As root (using `sudo`), you can manually load or unload the device's kernel module using the `modprobe` command along with the module's name. For example, use the following command line to enable the RealTek NIC from the earlier example:

```
matthew@seymour:~$ sudo modprobe 8139too
```

After you press Enter, you see this device reported from the kernel's ring buffer messages, which you can display by using the `dmesg` command. Here's a portion of that command's output:

```
matthew@seymour:~$ dmesg
```

```
...
eth0: RealTek RTL8139 Fast Ethernet at 0xce8ee000, 00:30:1b:0b:07:0d, IRQ 11
eth0: Identified 8139 chip type 'RTL-8139C'
eth0: Setting half-duplex based on auto-negotiated partner ability 0000.
...
```

Note that at this point, an IP address and other settings have not been assigned to the device. Linux can use multiple Ethernet interfaces, with the first Ethernet device numbered `eth0`, the second `eth1`, and so on. Each different Ethernet device recognized by the kernel might have additional or different information reported, depending on its kernel module. Here is an example:

```
matthew@seymour:~$ dmesg
...
eepro100.c:v1.09j-t 9/29/99 Donald Becker http://cesdis.gsfc.nasa.gov/linux/drivers/eepro100.html
eepro100.c: $Revision: 1.36 $ 2000/11/17 Modified by Andrey V. Savochkin
&#x194;<saw@saw.sw.com.sg> and others
PCI: Found IRQ 10 for device 00:0d.0
eth0: Intel Corporation 82557 [Ethernet Pro 100], 00:90:27:91:92:B5, IRQ 10.
Board assembly 721383-007, Physical connectors present: RJ45
Primary interface chip i82555 PHY #1.
General self-test: passed.
Serial sub-system self-test: passed.
Internal registers self-test: passed.
ROM checksum self-test: passed (0x04f4518b).
...
```

In this example, an Intel Ethernet Pro 100 NIC has been recognized. To disable support for a NIC, the kernel module can be unloaded, but usually only after the device is no longer in use. Read the next section to learn how to configure a NIC after it has been recognized by the Linux kernel and how to control its behavior.

Using Network Configuration Tools

If you add or replace networking hardware after initial installation, you must configure the new hardware. You can do so using either the command line or the graphical configuration tools. To configure a network client host using the command line, you can use a combination of commands or edit specific files under the `/etc` directory. To configure the hardware through a graphical interface, you can use Ubuntu's graphical tool for X called `nm-connection-editor`, found by clicking the Network indicator and then Edit Connections. This section introduces command-line and graphical software tools you can use to configure a network interface and network settings on your Ubuntu system. You'll see how to control your NIC and manage how your system interacts with your network.

Using the command-line configuration tools can seem difficult if you are new to Linux. For anyone new to networking, using the `nm-connection-editor` graphical tool is the way

to go. Both manual and graphical methods require super user privileges. You should not edit any scripts or settings files used by graphical network administration tools on your system, or your changes will be lost the next time the tool is run. Either use a manual approach all the time and write your own network setup script or stick to using graphical configuration utilities. Don't switch back and forth between the two methods.

Command-Line Network Interface Configuration

You can configure a network interface from the command line by using the basic Linux networking utilities. You configure your network client hosts either with commands to change your current settings or by editing a number of system files. Traditionally, two commands, `ifconfig` (which has generally been abandoned for `ip`, as mentioned earlier in this chapter) and `ip route`, are used for network configuration. The `netstat` command displays information about the network connections.

NOTE

`ifconfig` has been replaced by `ip`, which is also covered in this section. As you are likely to encounter older systems and admins still using `ifconfig`, information on this command has been retained here as well. Feel free to skip ahead to the `ip` section.

`ifconfig`

`ifconfig` is used to configure a network interface. You can use it to do the following:

- ▶ Activate or deactivate your NIC or change your NIC's mode
- ▶ Change your machine's IP address, netmask, or broadcast address
- ▶ Create an IP alias to allow more than one IP address on your NIC
- ▶ Set a destination address for a point-to-point connection

You can change as many or as few of these options as you want with a single command. The basic structure for the command is as follows:

```
ifconfig [network device] options
```

Table 18.1 shows a subset of `ifconfig` options and examples of their uses.

Table 18.1 `ifconfig` Options

Use	Option	Example
Create alias	<i>[network device]</i>	<code>ifconfig eth0:0 _:[number] 10.10.10.10</code>
Change IP address		<code>ifconfig eth0 10.10.10.12</code>
Change the netmask	<code>netmask [netmask]</code>	<code>ifconfig eth0 netmask 255.255.255.0</code>

Use	Option	Example
Change the broadcast	<code>broadcast [address]</code>	<code>ifconfig eth0 broadcast 10.10.10.255</code>
Take interface down	<code>down</code>	<code>ifconfig eth0 down</code>
Bring interface up	<code>up (add IP address)</code>	<code>ifconfig eth0 up (ifconfig eth0 10.10.10.10)</code>
Set NIC promiscuous	<code>[-]promisc</code> <code>[ifconfig eth0 -promisc]</code>	<code>ifconfig eth0 promisc mode on [off]</code>
Set multicasting mode	<code>[-]allmulti</code>	<code>ifconfig eth0 _on [off] allmulti [ifconfig eth0 -allmulti]</code>
Enable or disable	<code>[-]pointopoint [address]</code> <code>eth0 _pointopoint</code>	<code>ifconfig _point-to-point address 10.10.10.20 [ifconfig eth0 pointopoint _10.10.10.20]</code>

The `ifconfig` man page shows other options that enable your machine to interface with a number of network types, such as AppleTalk, Novell, IPv6, and others. Again, read the man page for details on these network types.

NOTE

Promiscuous mode causes the NIC to receive all packets on the network. It is often used to sniff a network. Multicasting mode enables the NIC to receive all multicast traffic on the network.

If no argument is given, `ifconfig` displays the status of active interfaces. For example, the output of `ifconfig`, without arguments and one active and configured NIC, looks similar to this:

```
matthew@seymour:~$ ifconfig
eth0      Link encap:Ethernet  HWaddr 00:90:f5:8e:52:b5
          UP BROADCAST MULTICAST  MTU:1500  Metric:1
          RX packets:0 errors:0 dropped:0 overruns:0 frame:0
          TX packets:0 errors:0 dropped:0 overruns:0 carrier:0
          collisions:0 txqueuelen:1000
          RX bytes:0 (0.0 B)  TX bytes:0 (0.0 B)
          Interrupt:30 Base address:0xc000
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:314 errors:0 dropped:0 overruns:0 frame:0
          TX packets:314 errors:0 dropped:0 overruns:0 carrier:0
          collisions:0 txqueuelen:0
          RX bytes:25204 (25.2 KB)  TX bytes:25204 (25.2 KB)
```

```
wlan0    Link encap:Ethernet  HWaddr 00:16:ea:d4:58:88
         inet addr:192.168.1.106  Bcast:192.168.1.255  Mask:255.255.255.0
         inet6 addr: fe80::216:eaff:fed4:5888/64 Scope:Link
         UP BROADCAST RUNNING MULTICAST  MTU:1500  Metric:1
         RX packets:325832 errors:0 dropped:0 overruns:0 frame:0
         TX packets:302754 errors:0 dropped:0 overruns:0 carrier:0
         collisions:0 txqueuelen:1000
         RX bytes:207381807 (207.3 MB)  TX bytes:40442735 (40.4 MB)
```

This output is easily understood. The `inet` entry displays the IP address for the interface. `UP` signifies that the interface is ready for use; `BROADCAST` denotes that the interface is connected to a network that supports broadcast messaging (`ethernet`); `RUNNING` means that the interface is operating; and `LOOPBACK` shows which device (`lo`) is the loopback address. The *maximum transmission unit (MTU)* on `eth0` is 1500 bytes. This determines the size of the largest packet that can be transmitted over this interface (and is sometimes “tuned” to other values for performance enhancement). `Metric` is a number from 0 to 3 that relates to how much information from the interface is placed in the routing table. The lower the number, the smaller the amount of information.

The `ifconfig` command can be used to display information about or control a specific interface using commands that are listed in Table 18.1. For example, to deactivate the first Ethernet device on a host, use the `ifconfig` command, the interface name, and the command `down`, like this:

```
matthew@seymour:~$ sudo ifconfig eth0 down
```

You can also configure and activate the device by specifying a hostname or an IP address and network information. For example, to configure and activate (bring up) the `eth0` interface with a specific IP address, use the `ifconfig` command as follows:

```
matthew@seymour:~$ sudo ifconfig eth0 192.168.2.9 netmask 255.255.255.0 up
```

If you have a host defined in your system’s `/etc/hosts` file (see the section “Network Configuration Files,” later in this chapter), you can configure and activate the interface according to the defined hostname, like this:

```
matthew@seymour:~$ sudo ifconfig eth0 catcat.fakeurl.com up
```

ip

In preparing for this edition, `ifconfig` still worked well on our testing system, but it was no longer installed by default. It is losing favor as `ip` sees more use. This command works with a series of subcommands to perform its tasks. Many of the common subcommands also have short aliases, which are also listed here. Note that the IP addresses listed next are examples; the addresses in your network will likely be different.

The following command allows you to get information about all your network interfaces:

```
matthew@seymour:~$ sudo ip addr show
```

To assign an IP address to a specific interface, in this case `eth1`, use the following command:

```
matthew@seymour:~$ sudo ip addr add 192.168.2.9 dev eth1
```

To remove an assigned IP address, use this:

```
matthew@seymour:~$ sudo ip addr del 192.168.2.9 dev eth1
```

Enable a network interface as shown here:

```
matthew@seymour:~$ sudo ip link set eth1 up
```

To disable a network interface, use the following:

```
matthew@seymour:~$ sudo ip link set eth1 down
```

Check the routing table as follows:

```
matthew@seymour:~$ sudo ip route show
```

To add a static route, do the following:

```
matthew@seymour:~$ sudo ip route add 10.10.30.0/24 via 192.168.50.100 dev eth0
```

Use the following command to remove a static route:

```
matthew@seymour:~$ sudo ip route del 10.10.30.0/24
```

To add a default gateway, use the following:

```
matthew@seymour:~$ sudo ip route add default via 192.168.36.100
```

The next section explains how to configure your system to work with your LAN.

ip route

Another command used to configure your network is the `ip route` command. `ip route` is used to build the routing tables (in memory) implemented for routing packets and to display the routing information. It is used after `ip` (or `ifconfig`) has initialized the interface. `ip route` is normally used to set up static routes to other networks via the gateway or to other hosts.

To display the current routing configuration, use the `ip route` command with no options. The display will look similar to this:

```
matthew@seymour:~$ ip route
default via 192.168.1.1 dev enp1s0 proto static metric 100
169.254.0.0/16 dev enp1s0 scope link metric 1000
192.168.1.0/24 dev enp1s0 proto kernel scope link src 192.168.1.148 metric 100
```

netstat

The `netstat` command is used to display the status of your network. It has several parameters that can display as much or as little information as you prefer. The services are listed by *sockets* (application-to-application connections between two computers). You can use `netstat` to display the information listed in Table 18.2.

Table 18.2 `netstat` Options

Option	Output
<code>-g</code>	Displays the multicast groups configured
<code>-i</code>	Displays the interfaces configured by <code>ifconfig</code>
<code>-s</code>	Lists a summary of activity for each protocol
<code>-v</code>	Gives verbose output, listing both active and inactive sockets
<code>-c</code>	Updates output every second (good for testing and troubleshooting)
<code>-e</code>	Gives verbose output for active connections only
<code>-C</code>	Displays information from the route cache and is good for looking at past connections

Several other options are available for this command, but they are used less often.

Network Configuration Files

As previously stated, five network configuration files can be modified to make changes to basic network interaction of your system:

- ▶ `/etc/hosts`—A listing of addresses, hostnames, and aliases
- ▶ `/etc/services`—Network service and port connections
- ▶ `/etc/nsswitch.conf`—Linux network information service configuration
- ▶ `/etc/resolv.conf`—*Domain Name System (DNS)* domain (search) settings
- ▶ `/etc/host.conf`—Network information search order (by default, `/etc/hosts` and then DNS)

After these files are modified, the changes are active. With most configuration files, you can add comments with a hash mark (#) preceding a comment. All these files have man pages where you can find more information.

Adding Hosts to `/etc/hosts`

The `/etc/hosts` file is a map of IP addresses to hostnames. If you are not using DNS or another naming service and you are connected to a large network, this file can get quite large, and managing it can be a real headache. A small `/etc/hosts` file can look something like this:

```
127.0.0.1    localhost
127.0.1.1    optimus
```

```
# The following lines are desirable for IPv6 capable hosts
::1      ip6-localhost ip6-loopback
fe00::0  ip6-localnet
ff00::0  ip6-mcastprefix
ff02::1  ip6-allnodes
ff02::2  ip6-allrouters
ff02::3  ip6-allhosts
```

The first entry is for the loopback entry. The second is for the name of the machine. If no naming service is in use on the network, the only host that `myhost` recognizes by name is `yourhost`. (IP addresses on the network can still be used.)

Service Settings in `/etc/services`

The `/etc/services` file maps port numbers to services. The first few lines of the file (which can be more than 500 lines long in total) look similar to this:

```
# Each line describes one service, and is of the form:
#
# service-name port/protocol [aliases ...] [# comment]

tcpmux      1/tcp          # TCP port service multiplexer
tcpmux      1/udp          # TCP port service multiplexer
rje         5/tcp          # Remote Job Entry
rje         5/udp          # Remote Job Entry
echo        7/tcp
echo        7/udp
discard     9/tcp          sink null
discard     9/udp          sink null
sysstat     11/tcp         users
```

Typically, there are two entries for each service because most services can use either TCP or UDP for their transmissions. Usually after `/etc/services` is initially configured, you do not need to change it.

Using `/etc/nsswitch.conf` After Changing Naming Services

The `/etc/nsswitch.conf` file was initially developed by Sun Microsystems to specify the order in which services are accessed on the system. A number of services are listed in the `/etc/nsswitch.conf` file, but the most commonly modified entry is the `hosts` entry. A portion of the file may look like this:

```
passwd:      compat
group:       compat
shadow:      compat

hosts:       files dns mdns
networks:    files
```

```

protocols:      db files
services:      db files
ethers:        db files
rpc:           db files

netgroup:      nis

```

This tells services that they should consult standard UNIX/Linux files for `passwd`, `shadow`, and `group` (`/etc/passwd`, `/etc/shadow`, `/etc/group`, respectively) lookups. For host lookups, the system checks `/etc/hosts`; if there is no entry, it checks DNS. The commented `hosts` entry lists the possible values for `hosts`. Edit this file only if your naming service has changed.

Setting a Name Server with `/etc/resolv.conf`

Note that this section is old, but included for those using older releases of Ubuntu. For releases newer than 17.10, see “Setting a Name Server with `/etc/netplan/*.yaml`.”

`/etc/resolv.conf` is used by DNS. The following is an example of `resolv.conf`:

```

nameserver 192.172.3.8
nameserver 192.172.3.9
search mydomain.com

```

This sets the `nameservers` and the order of domains for DNS to use. The contents of this file are set automatically if you use DHCP (see the “Dynamic Host Configuration Protocol” section, later in this chapter).

Starting with 12.04, there was a pretty big change in how Ubuntu uses the `/etc/resolv.conf` file. Management of `resolv.conf` has been turned over to a program called `resolvconf`, which works with DHCP, with a Network Manager plug-in, and with `/etc/network/interfaces` to automatically generate a list of nameservers and domains to list in `/etc/resolv.conf`. This means that any changes made here manually are eventually overwritten and lost.

If you have a static IP configuration, you should now list each of your static IP interfaces as `dns-nameservers`, `dns-search`, and `dns-domain` entries in `/etc/network/interfaces`.

You can override the configuration for `resolvconf` or add entries to it in the following files in the `/etc/resolvconf/resolv.conf.d/` directory:

- ▶ **base**—This file is used when no other data can be found.
- ▶ **head**—This file is used as the header for `resolv.conf`, and you can use it to ensure that a specific DNS server is always the first one on the list used.
- ▶ **original**—This file is a backup copy of your original `resolv.conf` file from the time when the `resolvconf` program was installed.
- ▶ **tail**—This file is used as a tail, appended to the end of the auto-generated `resolv.conf` file.

The format in these files is the same as the traditional format for `/etc/resolv.conf`. Splitting things this way allows for more granular control while also allowing for DHCP auto-configuration.

Setting a Name Server with `/etc/netplan/*.yaml`

Starting with 17.10, Ubuntu made another big switch away from `resolvconf` to Netplan. Instead of using `/etc/network/interfaces`, you now find network configuration in `/etc/netplan/*.yaml` files.

By default, only one file exists in this location on the desktop, `01-network-manager-all.yaml`, and `01-netcfg.yaml` if you installed using the server options. Either includes simple contents like these:

```
# Let NetworkManager manage all devices on this system
network:
  version: 2
  renderer: NetworkManager
```

YAML files are strict in their use of indentation, so pay attention when working with them.

This configures the first network interface (or only if you only have one). If you have multiple interfaces you will either find or must create additional files, incrementing the starting number in the filenames as you add interfaces, such as `02-netcfg.yaml`.

If you have a static IP address, start by finding the assigned name to each of your network interfaces, using `ip a` like this (your output will be different based on your hardware):

```
matthew@seymour:~$ ip a
1: lo: <LOOPBACK,UP,LOWER_UP> mtu 65536 qdisc noqueue state UNKNOWN group
default qlen 1000
    link/loopback 00:00:00:00:00:00 brd 00:00:00:00:00:00
    inet 127.0.0.1/8 scope host lo
        valid_lft forever preferred_lft forever
    inet6 ::1/128 scope host
        valid_lft forever preferred_lft forever
2: enp2s0: <NO-CARRIER,BROADCAST,MULTICAST,UP> mtu 1500 qdisc mq state DOWN
group default qlen 1000
    link/ether d8:d3:85:94:5d:3f brd ff:ff:ff:ff:ff:ff
3: enp1s0: <BROADCAST,MULTICAST,UP,LOWER_UP> mtu 1500 qdisc mq state UP group
default qlen 1000
    link/ether d8:d3:85:94:5d:3e brd ff:ff:ff:ff:ff:ff
    inet 192.168.1.149/24 brd 192.168.1.255 scope global dynamic noprefixroute
enp1s0
```

The IP address for each interface is listed in its entry. In this instance, there are two entries: One for `lo`, the loopback interface, and two others, `enp2s0` and `enp1s0`. These last two entries are what will be useful in the next step.

To set a nameserver, you add lines with information about them to create a longer file, like this:

```
network:
  Version: 2
  Renderer: NetworkManager/ networkd
  ethernets:
    DEVICE_NAME:
      Dhcp4: yes/no
      Addresses: [IP_ADDRESS/NETMASK]
      Gateway: GATEWAY
      Nameservers:
        Addresses: [NAMESERVER_1, NAMESERVER_2]
    DEVICE_NAME:
      Dhcp4: yes/no
      Addresses: [IP_ADDRESS/NETMASK]
      Gateway: GATEWAY
      Nameservers:
        Addresses: [NAMESERVER_1, NAMESERVER_2]
```

Next, you will add `DEVICE_NAME` entries for every network device you want to configure. Replace `DEVICE_NAME` with the interface from the previous step where we used `ip a` and create an additional section starting with this line for each interface.

In each section, set `Dhcp4` to either `yes` or `no` depending on whether the interface will use dynamic or static IP addressing. `Yes` means you are using dynamic.

Then set the `IP_ADDRESS` and `NETMASK` for each.

Finally, set the `NAMESERVERS` you want to use for each interface.

Save the file and test it using this:

```
matthew@seymour:~$ sudo netplan try
[sudo] password for matthew:
Do you want to keep these settings?
```

Press ENTER before the timeout to accept the new configuration

```
Changes will revert in 112 seconds
Configuration accepted.
```

If the test is successful and the configuration is accepted, apply the configuration like this:

```
matthew@seymour:~$ sudo netplan apply
```

If the test was unsuccessful, you can try to fix it yourself (some fixes are obvious like egregious typos, which we all make) or you can run the `debug` command and fix what it tells you:

```
matthew@seymour:~$ sudo netplan -d apply
```

Once the configurations are successfully applied, restart the Network Manager service using this on the desktop:

```
matthew@seymour:~$ sudo systemctl restart network-manager
```

Or this on a server:

```
matthew@seymour:~$ sudo systemctl restart system-networkd
```

Then use `ip a` again to verify your settings.

Setting DNS Search Order with `/etc/host.conf`

This section is only used by legacy services—which still exist—so don’t skip this section entirely as it may be useful on your system. Use the next section for modern services. It is a good idea to configure both options and do so identically.

The `/etc/host.conf` file lists the order in which your machine searches for hostname resolution. The following is the default `/etc/host.conf` file:

```
order hosts, bind
```

In this example, the host checks the `/etc/hosts` file first and then performs a DNS lookup. A couple more options control how the name service is used. The only reason to modify this file is if you use NIS for your name service or you want one of the optional services. The `nospoof` option can be a good option for system security. It compares a standard DNS lookup to a reverse lookup (host-to-IP and then IP-to-host) and fails if the two don’t match. The drawback is that often when proxy services are used, the lookup fails, so you should use this with caution.

Setting DNS Search Order with `/etc/nsswitch.conf`

This section is only used by modern services. Legacy services, which your system may still run, are configured using the previous section. It is a good idea to configure both options and do so identically.

The `/etc/nsswitch.conf` file lists the order in which your machine searches for hostname resolution in the `hosts:` line. Set the order in that line as needed and save the file.

Using Graphical Configuration Tools

Ubuntu provides options for desktop users to configure networking using graphical configuration tools. In most cases, all you need to know is contained in Chapter 1, “Installing Ubuntu and Post-Installation Configuration,” in the section “Configuring Wireless Networks.” Power users and unique setups generally eschew the GUI and use command-line tools.

Dynamic Host Configuration Protocol

As its name implies, *Dynamic Host Configuration Protocol (DHCP)* is used to configure hosts for connection to your network. DHCP enables a network administrator to configure all TCP/IP parameters for each host as she connects them to the network after activation of

a NIC. These parameters include automatically assigning an IP and DNS configuration to a NIC, setting name server entries in `/etc/resolv.conf`, and configuring default routing and gateway information for a host. This section first describes how to use DHCP to obtain IP address assignment for your NIC and then how to quickly set up and start a DHCP server using Ubuntu.

NOTE

You can learn more about DHCP by reading RFC 2131, “Dynamic Host Configuration Protocol,” at www.ietf.org/rfc/rfc2131.txt.

How DHCP Works

DHCP provides persistent storage of network parameters by holding identifying information for each network client that might connect to the network. The three most common pairs of identifying information are as follows:

- ▶ **Network subnet/host address**—Enables hosts to connect to the network at will
- ▶ **Subnet/hostname**—Enables the specified host to connect to the subnet
- ▶ **Subnet/hardware address**—Enables a specific client to connect to the network after getting the hostname from DHCP

DHCP also allocates to the client’s temporary or permanent network (IP) addresses. When a temporary assignment, known as a *lease*, elapses, the client can request to have the lease extended, or, if the address is no longer needed, the client can relinquish the address. For hosts that will be permanently connected to a network with adequate addresses available, DHCP allocates infinite leases.

DHCP offers your network some advantages. First, it shifts responsibility for assigning IP addresses from the network administrator (who can accidentally assign duplicate IP addresses) to the DHCP server. Second, DHCP makes better use of limited IP addresses. If a user is away from the office for whatever reason, the user’s host can release its IP address for use by other hosts.

Like most other things in life, DHCP is not perfect. Servers cannot be configured through DHCP alone because DNS does not know what addresses DHCP assigns to a host. This means that DNS lookups are not possible on machines configured through DHCP alone; therefore, services cannot be provided. However, DHCP can make assignments based on DNS entries when using subnet/hostname or subnet/hardware address identifiers.

NOTE

The problem of using DHCP to configure servers using registered hostnames is being addressed by Dynamic DNS, which, when fully developed, will enable DHCP to register IP addresses with DNS. This will enable you, for example, to register a domain name (such as `matthewhelmke.com`) and be able to easily access that domain’s web server without needing to use static IP addressing of a specific host. The largest hurdle to overcome is

the security implication of enabling each host connecting to the system to update DNS. A few companies, such as Dyn.com (www.dynDNS.org), are already offering Dynamic DNS services and have clients for Linux.

Activating DHCP at Installation and Boot Time

Ubuntu automatically defaults your network interfaces to using DHCP because it is the simplest way of setting up a network interface. With *dynamic*, or DHCP-assigned IP addressing schemes for your NIC, the broadcast address is set at 255.255.255.255 because `dhclient`, the DHCP client used for IP configuration, is initially unaware of where the DHCP server is located, so the request must travel every network until a server replies.

You can find the instruction to use DHCP for your NIC in `/etc/netplan/*.yaml`, in a line that says `dhcp`.

Other settings specific to obtaining DHCP settings are saved in the file `/etc/dhcp/dhclient.conf` and are documented in the `dhclient.conf` man page. More than 100 options are also documented in the `dhcpcoptions` man page.

However, using DHCP is not very complicated. If you want to use DHCP and know that there is a server on your network, you can quickly configure your NIC by using the `dhclient`, as follows:

```
matthew@seymour:~$ sudo dhclient
Internet Systems Consortium DHCP Client V3.1.3
Copyright 2004-2009 Internet Systems Consortium.
All rights reserved.
For info, please visit https://www.isc.org/software/dhcp/

Listening on LPF/eth0/00:90:f5:8e:52:b5
Sending on   LPF/eth0/00:90:f5:8e:52:b5
Listening on LPF/virbr0/ee:1a:62:7e:e2:a2
Sending on   LPF/virbr0/ee:1a:62:7e:e2:a2
Listening on LPF/wlan0/00:16:ea:d4:58:88
Sending on   LPF/wlan0/00:16:ea:d4:58:88
Sending on   Socket/fallback
DHCPDISCOVER on eth0 to 255.255.255.255 port 67 interval 7
DHCPDISCOVER on wlan0 to 255.255.255.255 port 67 interval 3
DHCPOFFER of 192.168.1.106 from 192.168.1.1
DHCPREQUEST of 192.168.1.106 on wlan0 to 255.255.255.255 port 67
DHCPACK of 192.168.1.106 from 192.168.1.1
bound to 192.168.1.106 -renewal in 35959 seconds.
```

In this example, the first Ethernet device, `eth0`, has been assigned IP address 192.168.1.106 from a DHCP server at 192.168.1.1. The renewal will take place in 35959 seconds, or about 10 hours. (Cool tip: Google converts this for you if you search for “35959 seconds in hours.”)

DHCP Software Installation and Configuration

Installation of the DHCP client and server is fairly straightforward, mainly because Ubuntu already includes `dhclient` in a default installation but also because installing software is easy using `synaptic` or `apt-get`.

DHCP `dhclient`

DHCP is automatically enabled when you install Ubuntu, so you do not need to worry about having to enable it. The DHCP client, `dhclient`, sends a broadcast message that the DHCP server replies to with networking information for your host. After it has this, you're done.

You can, however, fine-tune how `dhclient` works and where and how it obtains or looks for DHCP information. You probably will not need to take this additional effort, but if you do, you can create and edit a file named `dhclient.conf` and save it in the `/etc/dhcp` directory with your settings.

CAUTION

You should not just go ahead and overwrite your `dhclient.conf` with any old file because doing so could lead to painful networking problems. Instead, copy the file like this:

```
matthew@seymour:~$ sudo cp /etc/dhcp/dhclient.conf/etc/dhcp/dhclient.conf.backup
```

This way, if anything goes wrong, you can use the backup to restore the original settings by copying it back to its original location in place of the modified file.

A few of the `dhclient.conf` options include the following:

- ▶ **timeout time** ;—How long to wait before giving up trying. (The default is 60 seconds.)
- ▶ **retry time** ;—How long to wait before retrying. (The default is 5 minutes.)
- ▶ **select-timeout time** ;—How long to wait before selecting a DHCP offer. (The default is 0 seconds.)
- ▶ **reboot time** ;—How long to wait before trying to get a previously set IP address. (The default is 10 seconds.)
- ▶ **renew date** ;—When to renew an IP lease, where *date* is in the form *weekday year/month/day hour:minute:second*, such as `3 2018/7/11 22:01:01` for Wednesday, July 11, 2018, at 10:01 p.m.

See the `dhclient.conf` man page for more information on additional settings.

DHCP Server

The easiest way to install the DHCP server on your computer is to use either `synaptic` or `apt-get` to retrieve the `isc-dhcp-server` package. If you are so inclined, you can go to the *Internet Software Consortium (ISC)* website (www.isc.org) and download and build the source code yourself. However, we recommend that you stay with the package in the Ubuntu repositories because it will be easy to update if there are security updates.

If you decide to install from a source downloaded from the ISC website, the installation is straightforward. Just unpack your tar file, run `./configure` from the root of the source directory, run `make`, and finally, if there are no errors, run `make install`. This puts all the files used by the DHCP daemon in the correct places. If you have the disk space, it is best to leave the source files in place until you are sure that DHCP is running correctly; otherwise, you might delete the source tree.

NOTE

For whichever installation method you choose, be sure that a file called `/etc/dhcp/dhcpd.leases` is created. The file can be empty, but it does need to exist for `dhcpd` to start properly.

Using DHCP to Configure Network Hosts

Configuring your network with DHCP can look difficult but is actually easy if your needs are simple. The server configuration can take a bit of work depending on the complexity of your network and how much you want DHCP to do.

Configuring the server takes some thought and a little bit of work. Luckily, the work involves editing only a single configuration file, `/etc/dhcp/dhcpd.conf`. To start the server at boot time, use the `systemd enable` command.

The `/etc/dhcp3/dhcpd.conf` file contains all the information needed to run `dhcpd`. Ubuntu includes a sample `dhcpd.conf` file in `/usr/share/doc/dhcp*/dhcpd.conf.sample`. The DHCP server source files also contain a sample `dhcpd.conf` file.

You can think of the `/etc/dhcp/dhcpd.conf` file as a three-part file. The first part contains the following configurations for DHCP itself:

- ▶ **Setting the domain name**—`option domain-name "example.org"`
- ▶ **Setting DNS servers**—`option domain-name-servers ns1.example.org and ns2.example.org` (IP addresses can be substituted.)
- ▶ **Setting the default and maximum lease times**—`default-lease-time 3600 and max-lease-time 14400`

Other settings in the first part include whether the server is the primary (authoritative) server and what type of logging DHCP should use. These settings are considered defaults, and you can override them with the subnet and host portion of the configuration in more complex situations.

NOTE

The `dhcpd.conf` file requires a semicolon (`;`) after each command statement. If your configuration file has errors or runs improperly, check to make sure the semicolons appear where needed.

The next part of the `dhcpd.conf` file deals with the different subnets that your DHCP server serves; this section is quite straightforward. Each subnet is defined separately and can look like this:

```
subnet 10.5.5.0 netmask 255.255.255.224 {
    range 10.5.5.26 10.5.5.30;
    option domain-name-servers ns1.internal.example.org;
    option domain-name "internal.example.org";
    option routers 10.5.5.1;
    option broadcast-address 10.5.5.31;
    default-lease-time 600;
    max-lease-time 7200;
}
```

This defines the IP addressing for the 10.5.5.0 subnet. It defines the IP address range from 10.5.5.26 through 10.5.5.30 to be dynamically assigned to hosts that reside on that subnet. This example shows that you can set any TCP/IP option from the subnet portion of the configuration file. It shows which DNS server the subnet will connect to, which can be good for DNS server load balancing, or which can be used to limit the hosts that can be reached through DNS. It defines the domain name, so you can have more than one domain on your network. It can also change the default and maximum lease times.

If you want your server to ignore a specific subnet, you can do so as follows:

```
subnet 10.152.187.0 netmask 255.255.255.0 {
}
```

This defines no options for the 10.152.187.0 subnet; therefore, the DHCP server ignores it.

The last part of the `dhcpd.conf` file is for defining hosts. This can be good if you want a computer on your network to have a specific IP address or other information specific to that host. The key to completing the host section is to know the hardware address of the host. As you learned in the “Hardware Devices for Networking” section, earlier in this chapter, the hardware address is used to differentiate the host for configuration. You can obtain your hardware address by using the `ip` command, as described previously:

```
host hopper {
    hardware ethernet 08:00:07:26:c0:a5;
    fixed-address hopper.matthewhelmke.com;
}
```

This example takes the host with the hardware address `08:00:07:26:c0:a5` and does a DNS lookup to assign the IP address for `hopper.matthewhelmke.com` to the host.

DHCP can also define and configure booting for diskless clients, like this:

```
host bumblebee {
    hardware ethernet 0:0:c0:5d:bd:95;
    filename "vmunix.bumblebee";
}
```

```
server-name "kernigan.matthewhelmke.com";
}
```

The diskless host `bumblebee` gets its boot information from server `kernigan.matthewhelmke.com` and uses `vmunix.bumblebee` kernel. All other TCP/IP configuration can also be included.

CAUTION

Remember that, to avoid problems, only one DHCP server should be configured on a local network. DHCP might not work correctly for you on a LAN with hosts running outdated legacy operating systems. Often, Windows NT servers have the Windows DHCP server installed by default. Because there is no configuration file for NT to sort through, that DHCP server configures your host before the Linux server if both machines are on the same LAN. Check your NT servers for this situation and disable DHCP on the NT server; afterward, your other DHCP-enabled hosts should configure correctly. Also check to make sure there are no conflicts if you use a cable or DSL modem, *wireless access point (WAP)*, or other intelligent router on your LAN that can provide DHCP.

Other Uses for DHCP

A whole host of options can be used in `dhcpd.conf`: Entire books are dedicated to DHCP. The most comprehensive book is *The DHCP Handbook*, by Ralph Droms and Ted Lemon. You can define NIS domains, configure NetBIOS, set subnet masks, and define time servers or many other types of servers (to name a few of the DHCP options you can use). The preceding example gets your DHCP server and client up and running.

The DHCP server distribution contains an example of the `dhcpd.conf` file that you can use as a template for your network. The file shows a basic configuration that can get you started with explanations for the options used.

Wireless Networking

Linux has had support for wireless networking since the first standards were developed in the early 1990s. With computers getting smaller and smaller, the uses for wireless networking have increased; meanwhile, the transmission speeds are increasing all the time. There are several ways to create a wireless network. The following sections introduce you to several Linux commands you can use to initialize, configure, and manage wireless networking on your Ubuntu system.

Support for Wireless Networking in Ubuntu

The Linux kernel that ships with Ubuntu provides extensive support for wireless networking. Related wireless tools for configuring, managing, or displaying information about a wireless connection include the following:

- ▶ **`iwconfig`**—Sets the network name, encryption, transmission rate, and other features of a wireless network interface

- ▶ **iwlist**—Displays information about a wireless interface, such as rate, power level, or frequency used
- ▶ **iwpriv**—Sets optional features of a wireless network interface, such as roaming
- ▶ **iwspy**—Shows wireless statistics of a number of nodes

Support varies for wireless devices, but most modern (that is, post-2005) wireless devices should work with Ubuntu. In general, Linux wireless device software (usually in the form of a kernel module) supports the creation of an Ethernet device that can be managed using traditional interface tools such as `ifconfig`—with wireless features of the device managed by the various wireless software tools.

For example, when a wireless networking device is first recognized and initialized for use, the driver most likely reports a new device, like so:

```
zd1211rw 5-4:1.0: firmware version 4725

zd1211rw 5-4:1.0: zd1211b chip 050d:705c v4810 \
high 00-17-3f AL2230_RF pa0 G-ns

zd1211rw 5-4:1.0: eth2

usbcore: registered new interface driver zd1211rw
```

This output (from the `dmesg` command) shows that the `eth2` device has been reported. If DHCP is in use, the device should automatically join the nearest wireless subnet and be automatically assigned an IP address. If not, the next step is to use a wireless tool such as `iwconfig` to set various parameters of the wireless device. The `iwconfig` command, along with the device name (`eth2` in this example), shows the status:

```
matthew@seymour:~$ iwconfig eth2
eth2      IEEE 802.11b/g  ESSID:"SKY35120"  Nickname:"zd1211"
          Mode:Managed  Frequency:2.462 GHz  \
Access Point: 00:18:4D:06:8E:2A
          Bit Rate=24 Mb/s
          Encryption key:0EFD-C1AF-5C8D-B2C6-7A89-3790-07A7-AC64-0AB5\
-C36E-D1E9-A230-1DB9-D227-2EB6-D6C8  Security mode:open
          Link Quality=100/100  Signal level=82/100
          Rx invalid nwid:0  Rx invalid crypt:0  Rx invalid frag:0
          Tx excessive retries:0  Invalid misc:0  Missed beacon:0
```

This example shows a 24Mbps connection to a network named `SKY35120`. To change a parameter, such as the transmission rate, use a command-line option with the `iwconfig` command like this:

```
matthew@seymour:~$ sudo iwconfig eth2 rate 11M
```

Other options supported by the `iwconfig` command include `essid`, to set the NIC to connect to a specific network by name; `mode`, to enable the NIC to automatically retrieve settings from an access point or connect to another wireless host; and `freq`, to set a frequency to use for communication. Additional options include `channel`, `frag`, `enc` (for encryption), `power`, and `txpower`. Details and examples of these options are in the `iwconfig` man page.

You can then use the `ifconfig` command or perhaps a graphical Ubuntu tool to set the device networking parameters, and the interface will work as on a hardwired LAN. One handy output of the `iwconfig` command is the link quality output, which you can use in shell scripts or other graphical utilities for signal-monitoring purposes.

Choosing from Among Available Wireless Protocols

The *Institute of Electrical and Electronics Engineers (IEEE)* started to look seriously at wireless networking in 1990. This is when the 802.11 standard was first introduced by the Wireless Local Area Networks Standards Working Group. The group based the standard roughly around the architecture used in cellular phone networks. A wireless network is controlled by a base station, which can be just a transmitter attached to the network or, more commonly these days, a router.

A larger network may use more than one base station. Networks with more than one base station are usually referred to as *distribution systems*. You can use a distribution system to increase coverage area and support roaming of wireless hosts. You can also use external omnidirectional antennas to increase the coverage area or, if required, you can use point-to-point or directional antennas to connect distant computers or networks.

The 802.11 standard specifies that wireless devices use a frequency range of 2400MHz to 2483.5MHz. This is the standard used in North America and Europe. In Japan, however, wireless networks are limited to a frequency range of 2471MHz to 2479MHz. Within these ranges, each network is given up to 79 nonoverlapping frequency channels to use. This reduces the chance of two closely located wireless networks using the same channel at the same time. It also allows for channel hopping, which can be used for security.

Beyond the Network and onto the Internet

Ubuntu supports Internet connections and the use of Internet resources in many different ways. You will find a wealth of Internet-related software included with this book's version of Ubuntu, and you can download hundreds of additional free utilities from a variety of sources. To use them, you must have a working Internet connection.

In this section, you learn how to set up an Internet connection in Ubuntu using a modem and *Point-to-Point Protocol (PPP)* as well as other connection methods, including *digital subscriber line (DSL)* and cable modem services. Just a few years ago, getting a dial-up connection working was difficult—hence, an entire chapter of this book was devoted to it. Today, as long as you have a hardware modem, dial-up configuration is simple, although this is quite rare these days. The Ubuntu developers and the wider Linux community have made great progress in making connectivity easier.

Although many experienced Linux users continue to use manual scripts to establish Internet connectivity, new users and experienced system administrators alike will find Ubuntu's graphical network configuration interface much easier to use. You learn how to use the Internet Connection Wizard in this chapter and how to configure Ubuntu to provide dial-in PPP support. The chapter also describes how to use Roaring Penguin's DSL utilities to manage connectivity through a cable modem connection.

Common Configuration Information

Although Ubuntu enables great flexibility in configuring Internet connections, that flexibility comes at the price of increased complexity. To configure Internet connectivity in Ubuntu, you must know more about the details of the connection process than you can learn from the information typically provided by your Internet service provider (ISP). In this section, you learn what to ask about and how to use the information.

Some ISPs are unaware of Linux or unwilling to support its use with their services. Fortunately, that attitude is rapidly changing, and the majority of ISPs offer services using standard protocols that are compatible with Linux, even if they (or their technical support people) aren't aware that their own ISPs are Linux friendly. You just need to press a little for the information you require.

If you are one of the few remaining people using a dial-up modem account (referred to in Linux as *PPP*, for the Point-to-Point Protocol it uses), your ISP will provide your computer with a static or dynamic IP address. A dynamic IP address changes each time you connect, whereas a static IP address remains the same. The ISP also might automatically provide your computer with the names of the DNS servers. You need to know the telephone number that your computer will dial in to for making the connection; your ISP supplies that number, too. You also need a working modem and need to know the device name of the modem (usually `/dev/modem`).

NOTE

Most IP addresses are dynamically assigned by ISPs. An ISP has a pool of addresses, and you get whatever address is available. From the ISP's viewpoint, a small number of addresses can serve a large number of people because not everyone will be online at the same time. For most Internet services, a dynamic IP address works well because it is the ISP's job to route that information to you, and it sits in the middle—between you and the service you want to use. But a dynamic IP address changes, and if someone needs to find you at the same address (if you run a website or a file transfer site, for example), an IP that changes every time you log on does not work well. For that, you need a static IP address. Because your ISP cannot reuse that IP address for its other customers, it will likely charge you more for a static IP address than for a dynamic one. Average consumers do not need the benefit of a static IP address and so are happy paying less for a dynamically assigned IP. Also, an ISP can provide DNS information automatically, thanks to DHCP.

If you are using DSL access or a cable modem, you might have a dynamic IP address provided through DHCP, or you might be assigned a static IP address. You might

automatically be provided with the names of the DNS servers if you use DHCP, or you might have to set up DNS manually (in which case you have to know the IP addresses of the DNS servers).

In all cases, you have to know your username, your password, and, for the configuration of other services, the names of the mail servers and the news server. You can obtain this information from your ISP by specifically asking for it.

NOTE

The information in this book helps you understand and avoid many connection issues, but you might experience connection problems. Keep the telephone number of the technical help service for your ISP on hand in case you cannot establish a connection. But be aware that few ISPs offer Linux support, and you might need to seek help from a Linux-savvy friend or a Linux user group if your special circumstances cannot be handled from the knowledge you gain from this book. Of course, the best place to look is on the Internet.

Configuring Digital Subscriber Line Access

Ubuntu also supports the use of a *digital subscriber line (DSL)* service. Ubuntu refers to the different types of DSL available as *xDSL* (which includes ADSL, IDSL, SDSL, and other flavors of DSL service), and you can configure all of them by using the Internet Connection Wizard. DSL service generally provides 256Kbps to 24Mbps transfer speeds and transmits data over copper telephone lines from a central office to individual subscriber sites (such as your home). Many DSL services (technically, cable rather than DSL) provide asymmetric speeds with download speeds greater than upload speeds.

NOTE

DSL service is an “always-on” type of Internet service, although you can turn off the connection under Ubuntu by using the network configuration tool found under System, Administration, Network. An always-on connection exposes your computer to malicious abuse from crackers who trawl the Internet attempting to gain access to other computer systems. In addition to providing the capability to turn off such connections, Ubuntu is preconfigured not to listen on any network ports, which means that any attempts to gain access to your computer fail because Ubuntu rejects the requests. This is the Ubuntu equivalent to surrounding your computer with a 12-foot steel fence.

A DSL connection requires that you have an Ethernet NIC (sometimes a USB interface that is not easily supported in Linux) in your computer or notebook. Many users also configure a gateway, firewall, or other computer with at least two NICs to share a connection with a LAN. We looked at the hardware and protocol issues earlier in this chapter. Advanced configuration of a firewall or router, other than what was addressed during your initial installation of Ubuntu, is beyond the scope of this book.

Understanding PPP over Ethernet

Establishing a DSL connection with an ISP providing a static IP address is easy. Unfortunately, many DSL providers use a type of PPP called *Point-to-Point Protocol over Ethernet (PPPoE)* that provides dynamic IP address assignment and authentication by encapsulating PPP information inside Ethernet frames. Roaring Penguin's `rp-pppoe` clients are available from the Roaring Penguin site (www.roaringpenguin.com/files/download/rp-pppoe-3.11.tar.gz), and these clients make the difficult-to-configure PPPoE connection much easier to deal with. You can download and install newer versions.

NOTE

When ISPs originally started to roll out ADSL services, they often provided the ADSL modems. Today, however, in much of the world these modems are optional, which is a good thing because many people choose to purchase a router with a built-in modem to create a dedicated connection. In the United States, ADSL modems are rare, but you can usually replace the supplied modem with an aftermarket modem if you want to spend the money. Either way, using a router can save many headaches and enables you to easily connect more than one computer to an Internet connection. Note that a cable connection usually comes with an Ethernet cable, in which case you just need a router. Check with your ISP before buying to ensure that whatever router you end up with can be supported. You might find that your ISP even supplies a router as part of the package.

Configuring a PPPoE Connection Manually

You should need to use the steps in this section only if you are using a modem supplied by your ISP and not a router. The basic steps involved in manually setting up a DSL connection using Ubuntu involve connecting the proper hardware and then running a simple configuration script if you use `rp-pppoe` from Roaring Penguin.

First, connect your DSL modem to your telephone line and then plug in your Ethernet cable from the modem to your computer's NIC. If you plan to share your DSL connection with the rest of your LAN, you need at least two network cards, designated `eth0` (for your LAN) and `eth1` (for the DSL connection).

The following example assumes that you have more than one computer and will share your DSL connection on a LAN.

First, log in as root and ensure that your first `eth0` device is enabled and up (perhaps using the `ifconfig` command). Next, bring up the other interface but assign a null IP address like this:

```
matthew@seymour:~$ sudo ifconfig eth1 0.0.0.0 up
```

Now use the `adsl-setup` command to set up your system, as follows:

```
matthew@seymour:~$ sudo /sbin/adsl-setup
```

You are presented with a text script and asked to enter your username and the Ethernet interface used for the connection (such as `eth1`). You are then asked to use “on-demand”

service or have the connection stay up all the time (until brought down by the root operator). You can also set a timeout, in seconds, if desired. You are then asked to enter the IP addresses of your ISP's DNS servers if you haven't configured the DNS information on the network interface.

After that, you are prompted to enter your password two times and must choose the type of firewall and IP masquerading to use. (You learned about IP masquerading in the "Using IP Masquerading in Ubuntu" section, earlier in this chapter.) The actual configuration is done automatically. Using a firewall is essential, so choose this option unless you intend to craft your own set of firewall rules (a discussion of which is beyond the scope of this book). After you have chosen your firewall and IP masquerading setup, you are asked to confirm, save, and implement your settings. You are also given a choice to allow users to manage the connection, which is a handy option for home users.

Changes are made to your system's `/etc/netplan/*.yaml`, `/etc/ppp/pap-secrets`, and `/etc/ppp/chap-secrets` files.

After configuration has finished, use the `adsl-start` command to start a connection and DSL session, like this:

```
matthew@seymour:~$ sudo /sbin/adsl-start
```

The DSL connection should be nearly instantaneous, but if problems occur, check to make sure that your DSL modem is communicating with the phone company's central office by examining the status LEDs on the modem. This varies from modem to modem, so consult your modem user's manual.

Make sure all cables are properly attached, that your interfaces are properly configured, and that you have entered the correct information to the setup script.

If IP masquerading is enabled, other computers on your LAN on the same subnet address (such as `192.168.0.XXX`) can use the Internet but must have the same name server entries and a routing entry with the DSL-connected computer as a gateway. For example, if the host computer with the DSL connection has IP address `192.168.0.1`, and other computers on your LAN use addresses in the `192.168.0.XXX` range, use the `route` command on each computer like this:

```
matthew@seymour:~$ sudo route add default gw 192.168.0.1
```

Note that you can also use a hostname instead if each computer has an `/etc/hosts` file with hostname and IP address entries for your LAN. To stop your connection, use the `adsl-stop` command:

```
matthew@seymour:~$ sudo /sbin/adsl-stop
```

Configuring Dial-up Internet Access

Most ISPs provide dial-up connections that support PPP because it is a fast and efficient protocol for using TCP/IP over serial lines. PPP is designed for two-way networking, and

TCP/IP provides the transport protocol for data. One hurdle faced by new Ubuntu users is how to set up PPP and connect to the Internet. Understanding the details of the PPP protocol is not necessary to use it, and setting up a PPP connection is easy. You can configure PPP connections manually by using the command line or graphically during an X session using Ubuntu's Network Configuration tool. These approaches produce the same results.

PPP uses several components on your system. The first is a daemon called `pppd`, which controls the use of PPP. The second is a driver called the *high-level data link control (HDLC)*, which controls the flow of information between two machines. A third component of PPP is a routine called *chat* that dials the other end of the connection for you when you want it to. Although PPP has many “tunable” parameters, the default settings work well for most people.

Ubuntu includes some useful utilities to get your dial-up connection up and running. In this section, we look at two options that will have you on the Internet in no time.

The first way is to configure a connection using `pppconfig`, a command-line utility to help you configure specific dial-up connection settings.

Enter the following command:

```
matthew@seymour:~$ sudo pppconfig
```

Before you connect for the first time, you need to add yourself to both the `dip` and `dialout` groups by using these commands:

```
matthew@seymour:~$ sudo adduser YOURNAMEHERE dip
matthew@seymour:~$ sudo adduser YOURNAMEHERE dialout
```

After you have done this, it is just a simple matter of issuing the `pon` command to connect and the `pooff` command to disconnect. You can create as many different profiles as you need and can launch specific ones by using the command `pon profilename`, again using the `pooff` command to disconnect.

CAUTION

Many software modems do not work with Linux because the manufacturers do not release programming information about them or provide Linux drivers. External serial port modems or ISA bus modems almost always work; USB and PCI modems are still problematic. It is suggested that you do a thorough Google search, using your modem's name and model number, to see how others have solved problems with a particular modem. Links to software modem compatibility sites appear at the end of this chapter.

Troubleshooting Connection Problems

The Linux Documentation Project (www.tldp.org) offers many in-depth resources for configuring and troubleshooting connection problems. Google is also an invaluable tool for dealing with specific questions about connections. For many other useful references, see the “References” section at the end of this chapter.

Here are a few troubleshooting tips culled from many years of experience:

- ▶ If your modem connects and then hangs up, you are probably using the wrong password or dialing the wrong number. If the password and phone number are correct, it is likely an authentication protocol problem.
- ▶ If you get connected but cannot reach websites, it is likely a domain name resolver problem, meaning that DNS is not working. If it worked yesterday and you haven't "adjusted" the associated files, it is probably a problem on the ISP's end. Call and ask.
- ▶ Always make certain that everything is plugged in. Check again (and again).
- ▶ If the modem works in Windows but not in Linux no matter what you do, it is probably a software modem no matter what it said on the box.
- ▶ If everything just stops working (and you do not see smoke), it is probably a glitch at the ISP or the telephone company. Take a break and give them some time to fix it.
- ▶ Never configure a network connection when you have had too little sleep or too much caffeine; you will just have to redo it tomorrow.

RELATED UBUNTU AND LINUX COMMANDS

Use these commands when managing network connectivity in your Ubuntu system:

- ▶ **dhclient**—A tool for automatically acquiring and setting IP info for a NIC
- ▶ **ethereal**—The GNOME graphical network scanner
- ▶ **ufw**—Ubuntu's basic firewalling tool
- ▶ **ifconfig**—A tool that displays and manages Linux networking devices
- ▶ **iwconfig**—A tool that displays and sets wireless network device parameters
- ▶ **route**—A tool that displays and manages Linux kernel routing table
- ▶ **ssh**—The OpenSSH remote-login client and preferred replacement for `telnet`
- ▶ **nm-connection-editor**—Ubuntu's GUI for configuring network connections

References

- ▶ **<https://help.ubuntu.com/20.04/serverguide/networking.html>**—Official networking help for Ubuntu
- ▶ **www.ietf.org/rfc.html**—Site to search for or get a list of Requests for Comments (RFCs)
- ▶ **www.isc.org/products/DHCP/**—The official site for DHCP
- ▶ **www.ieee.org**—The Institute of Electrical and Electronics Engineers (IEEE) website

- ▶ ***Teach Yourself TCP/IP Network Administration in 21 Days* by Joe Casad**—Good introduction to the topic
- ▶ ***TCP/IP Network Administration* by Craig Hunt and Gigi Estabrook**—A more detailed look
- ▶ ***The DHCP Handbook* by Ralph Droms and Ted Lemon**—A complete reference for understanding DHCP

CHAPTER 19

Remote Access with SSH and VNC

Controlling your system remotely is one of the cooler things you can do with Ubuntu: With just a bit of configuration, you can connect from any Linux box to another computer in a variety of ways. If you just want to check something quickly or if you have limited bandwidth, you have the option of using only the command line, but you can also connect directly to the X server and get full graphical control.

Understanding the selection of tools available largely involves a history lesson. For example, Telnet was an earlier tool for connecting to another computer through the command line, but it has since been superseded by *Secure Shell* (SSH). For that reason, we will ignore Telnet as it is insecure for use, even as a fallback. This chapter covers SSH as well as *Virtual Network Computing* (VNC).

Setting Up an SSH Server

If SSH is not installed already, you can install the OpenSSH server by adding the `openssh-server` package. As you might have guessed, `sshd` is the name for the SSH server daemon.

Configure your firewall to allow connections through port 22. If this port is blocked, you cannot use SSH. See Chapter 20, “Securing Your Machines,” for help doing this.

Two different versions of SSH exist: SSH1 and SSH2. The latter is newer and more secure, comes with more features, and is the default in Ubuntu. Support for SSH1 clients is best left disabled so that older clients cannot connect. This is done by default in the `/etc/ssh/sshd_config` file, on this line:

```
Protocol 2
```

IN THIS CHAPTER

- ▶ Setting Up an SSH Server
- ▶ SSH Tools
- ▶ Virtual Network Computing
- ▶ Guacamole
- ▶ References

If you have no other option and absolutely have to allow an older client to connect, add this new line:

```
Protocol 1
```

SSH Tools

Many people are surprised to learn that OpenSSH actually is composed of a suite of tools. You have already seen `ssh`, the Secure Shell command that connects to other machines, and `sshd`, the SSH server daemon that accepts incoming SSH connections. However, there is also `sftp`, a replacement for `ftp`, `scp`, and `rcp`.

You may already be familiar with the `ftp` command because it is the lowest common denominator system for handling FTP file transfers. Like Telnet, though, `ftp` is insecure: It sends your data in plain text across the network, and anyone can sniff your packets to pick out a username and password. Don't use FTP. The SSH replacement, `sftp`, puts FTP traffic over an SSH link, thus securing it, which is safe and just as easy to use.

The `rcp` command might be new to you, largely because it is not used much anymore. Back in its day, `rcp` was the primary command for copying a single file to another server. As with `sftp` and `ftp`, `scp` replaces `rcp` by simply channeling the data over a secure SSH connection. The difference between `sftp` and `scp` is that the former enables you to queue and copy many files simultaneously, whereas the latter is usually used to send just one, although `scp` can be used with the `-r` option to send an entire directory at once. See the man page for details.

Using `scp` to Copy Individual Files Between Machines

The most basic use of the `scp` command is to copy a file from your current machine to a remote machine. You can do that with the following command:

```
matthew@seymour:~$ scp test.txt 192.168.1.102:
```

The first parameter is the name of the file you want to send, and the second is the server to which you want to send it. Note that there is a colon (:) at the end of the IP address. This is where you can specify an exact location for the file to be copied. If you have nothing after the colon, as in the previous example, `scp` copies the file to your `/home` directory. As with SSH, `scp` prompts you for your password before copying takes place.

You can rewrite the previous command so that you copy `test.txt` from the local machine and save it as `newtest.txt` on the server:

```
matthew@seymour:~$ scp test.txt 192.168.1.102:newtest.txt
```

Alternatively, if there is a directory where you want the file to be saved, you can specify it like this:

```
matthew@seymour:~$ scp test.txt 192.168.1.102:subdir/stuff/newtest.txt
```

The three commands so far have all assumed that your username on your local machine is the same as your username on the remote machine. If this is not the case, you need to specify your username before the remote address, as follows:

```
matthew@seymour:~$ scp test.txt username newtest.txt
```

You can use `scp` to copy remote files locally simply by specifying the remote file as the source and the current directory (`.`) as the destination:

```
matthew@seymour:~$ scp 192.168.1.102:remote.txt .
```

If you want to copy files from one remote machine to another remote machine using `scp`, the best method is to first `ssh` to one of the remote machines and then use `scp` from that location.

Using `sftp` to Copy Many Files Between Machines

`sftp` is a mix between `ftp` and `scp`. Connecting to the server requires the same syntax as with `scp`—you can just specify an IP address to connect using your current username, or you can specify a username by using `username@ipaddress`. You can optionally add a colon and a directory, as with `scp`. When you are connected, the commands are the same as for `ftp`: `cd`, `put`, `mput`, `get`, `quit`, and so on.

One of the `scp` examples earlier in this chapter copied a remote file locally. You can do the same thing with `sftp`, as shown in the following conversation:

```
matthew@seymour:~$ sftp 192.168.1.102
Connecting to 192.168.1.102...
matthew@192.168.1.102's password:
sftp> get remote.txt
Fetching /home/matthew/remote.txt to remote.txt
/home/matthew/remote.txt      100% 23  0.0KB/s   00:00
sftp> quit
matthew@seymour:~$
```

SFTP is popular because the initial connection between the client and server is made over port 22 through the `sshd` daemon. Someone using SFTP connects to the standard `sshd` daemon, verifies himself or herself, and then is handed over to the SFTP server. The advantage to this is that it reduces the attack vectors because the SFTP server cannot be contacted directly and so cannot be attacked as long as the `sshd` daemon is secure.

Using `ssh-keygen` to Enable Key-Based Logins

There is a weak link in the SSH system, and, inevitably, it lies with users. No matter what lengths system administrators go to in training users to be careful with their passwords, monitors around the world have sticky notes attached to them with `password` written on them. Sure, it has a mix of letters and numbers, but it can be cracked in less than a second

by any brute-force method. *Brute-forcing* is the method of trying every password possibility, starting with likely words (such as *password* and variants, or *god*) and then just trying random letters (for example, *a*, *aa*, *ab*, *ac*, and so on).

Even very strong passwords are no more than about 16 characters; such passwords take a long time to brute-force but can still be cracked. The solution is to use key-based logins; each such login generates a unique 1,024-bit private and public key pair for your machine. These keys take even the fastest computers a lifetime to crack, and you can back them up with a password to stop others from using them.

You create an SSH key by using the `ssh-keygen` command, like this:

```
matthew@seymour:~$ ssh-keygen -t dsa
```

Press Enter when the system asks you where to save your key, and enter a passphrase when it asks you to. This passphrase is just a password used to protect the key. You can leave it blank if you want to, but doing so will allow other people to use your account to connect to remote machines if they can manage to log in as you.

After the key is generated, change the directory to `.ssh` (`cd ~/.ssh`), which is a hidden directory where your key is stored and that keeps a list of safe SSH hosts. Assuming that you use the default options, here you see the files `id_dsa` and `id_dsa.pub`. The first is your private key, and you should never give it out. The second is your public key, which can safely be distributed. You need to copy the public key to each server you want to connect to via key-based SSH.

Use `ssh-copy-id` to copy the public key over to your server, like this:

```
matthew@seymour:~$ ssh-copy-id -i ~/.ssh/mykey user@host
```

The `touch` command creates the `authorized_keys` file (if it does not exist already); then you use `cat` to append the contents of `id_dsa.pub` to the list of already authorized keys. Finally, you use `chmod` to make `authorized_keys` read-only.

With that done, you can type `exit` to disconnect from the remote machine and return to your local machine. Then you can try running `ssh` again. If you are prompted for your passphrase, you have successfully configured key-based authentication.

If you have done all this, you have secured the current machine. But what about every other machine? Logging in from another machine using only a password is still possible, which means your remote machine is still vulnerable.

The solution to this is to edit the `/etc/ssh/sshd_config` file, which requires super user privileges. Look for the `PasswordAuthentication` line and make sure it reads `no` (and that it is not commented out with a `#`). Save the file and run `kill -HUP 'cat /var/run/sshd.pid'` to have `sshd` reread its configuration files. With that done, `sshd` accepts only connections from clients with authorized keys, which stops crackers from brute-forcing their way in.

TIP

For extra security, you are strongly encouraged to set `PermitRootLogin` to `no` in `/etc/ssh/sshd_config`. When this is set, it becomes impossible to `ssh` into your machine using the root account; you must connect with a normal user account and then use `su` or `sudo` to switch to root. This is advantageous because most brute-force attempts take place on the root account because it is the only account that is guaranteed to exist on a server.

Also, even if a cracker knows your user account, she has to guess both your user password and your root password to take control of your system.

Of course, if you don't have a root account enabled on your box, this isn't an issue because logging in directly as root is already impossible. Hooray for slightly more secure defaults in Ubuntu!

Virtual Network Computing

Everything we have looked at so far in this chapter has been about remote access using the command line, with no mention so far of how to bring up a *graphical user interface* (GUI). There are several ways of doing this in Ubuntu, some of which are listed in the “References” section, later in this chapter. Here we discuss the GUI that is most popular with and most preferred by the Ubuntu community and is also the GUI best supported by the developers: *Virtual Network Computing* (VNC).

VNC is a system for controlling a computer remotely and sharing desktops over a network using a graphical interface. It was created at the Olivetti & Oracle Research Lab, which was acquired by AT&T in 1999 and closed down in 2001. Later, several of the members of the development team got together to create a free and open source version of the code (using the GPL), and from that base, many different versions have appeared. VNC is widespread in the Linux world and, to a lesser extent, in the Windows world. Its main advantage is its widespread nature: Nearly all Linux distros bundle VNC, and clients are available for a wide selection of platforms.

Configure your firewall to allow connections through port 5900. If this port is blocked, you cannot use VNC.

To set up VNC on your Ubuntu computer to allow others to access your desktop, you need to tell Ubuntu who should be allowed to connect to your session. By default, your desktop is not shared. Open Settings, select Sharing from the menu on the left (see Figure 19.1), and then change the setting at the upper right from Off to On.

To access a computer that is running VNC from your Ubuntu desktop, run the Remmina Remote Desktop Client. The program starts by searching the local network for shared desktops. If any are found, they are listed. For example, Figure 19.2 shows Sandra's computer.

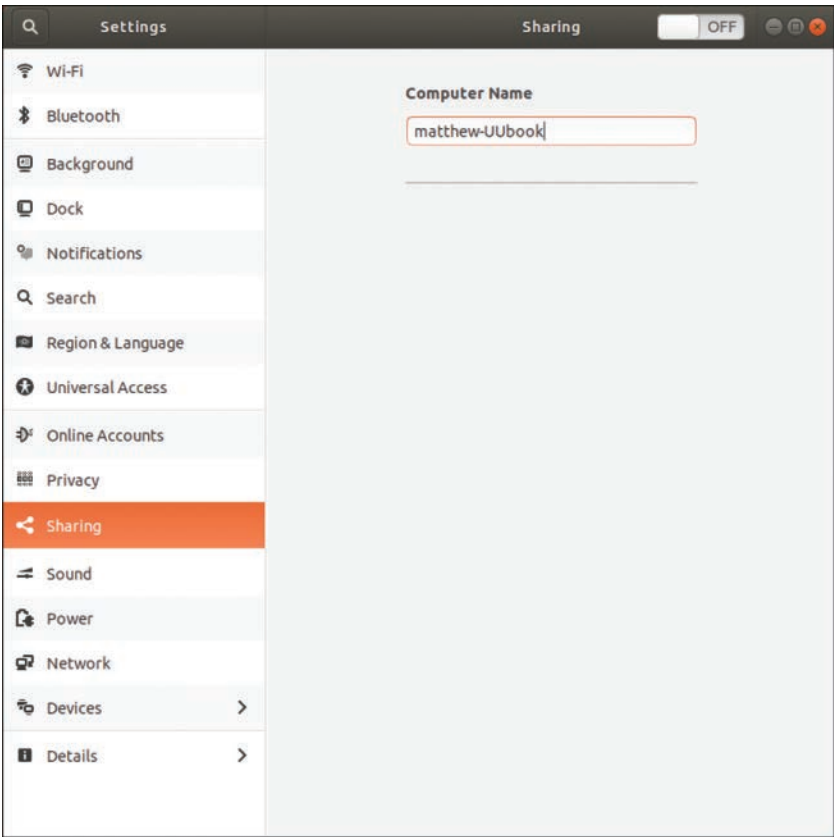


FIGURE 19.1 Desktop sharing preferences.

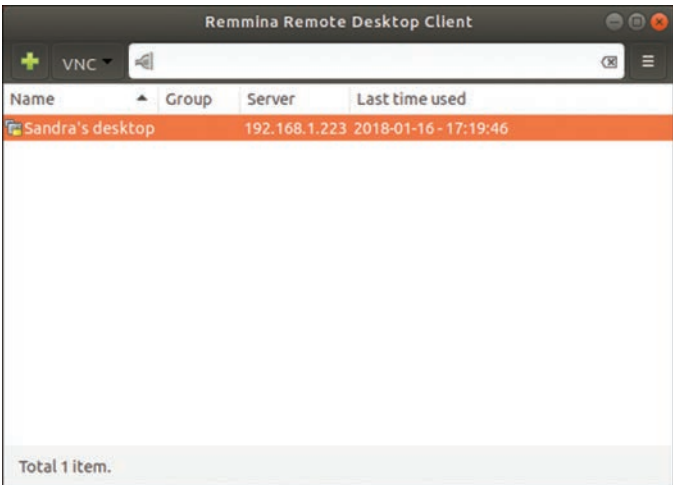


FIGURE 19.2 Remote desktop viewer.

From the screen shown in Figure 19.2, select an entry and select the Open the Connection icon at the top left, in the bar under the title bar with the window controls for close, minimize, and expand to full screen. If you have not yet entered any computers, select the + icon and enter the details for a computer you have already set up somewhere to be accessed.

When you have finished, close the window in which you are viewing the remote desktop. VNC should not be considered secure over the Internet or on untrusted internal networks.

Guacamole

Apache Guacamole is an Apache Software Foundation application that works as a clientless remote gateway using standard protocols, such as VNC, RDP, and SSH. It is called clientless because no plugins or client software are required. Guacamole is installed on a server, wherever you like as long as it is Internet-accessible, and then you access your desktop using a web browser.

The trick here is that your desktop doesn't actually exist. You use a host computer, but only for the web browser. The web browser connects to your Guacamole server, which hosts a virtual desktop in the cloud. When you use Guacamole, you can access your cloud-hosted desktop from anywhere, which is a pretty cool idea. Use of your computers is not tied to any specific device or location.

There are multiple ways to install Guacamole on your server, including both a typical native bare-metal installation and also a containerized Docker version. Authentication to Guacamole can be configured many ways, including using a local database, LDAP (Lightweight Directory Access Protocol), or via a single sign-on solution like CAS (Central Authentication Service).

The main use cases for Guacamole seems to be for business needs that want centralization and end-user customization of desktops for productivity while only maintaining a set of basic physical hardware to use as an interface. It also looks convenient for travelers and people who work remotely and move around frequently.

Guacamole was included in this chapter mainly for its unique use of technologies like VNC and SSH. If it sounds like something you may want to investigate further, you can learn more about Guacamole at <https://guacamole.apache.org/>.

References

- ▶ **www.telnet.org**—Telnet's home page
- ▶ **<https://help.ubuntu.com/community/SSH>**—The Ubuntu community documentation for SSH
- ▶ **www.openssh.com**—The home page of the OpenSSH implementation of SSH that Ubuntu uses, which is run by the same team as OpenBSD, a secure BSD-based operating system

- ▶ **<https://help.ubuntu.com/community/VNC>**—The Ubuntu community documentation for VNC
- ▶ **www.realvnc.com**—The home page of the team that made VNC at AT&T's Cambridge Research Laboratory and has since started RealVNC Ltd., a company dedicated to developing and supporting VNC
- ▶ **www.tightvnc.com**—An alternative to VNC called TightVNC that has several key advantages over the stock VNC release, including use of SSH for encryption, which guarantees security
- ▶ **<https://help.ubuntu.com/community/FreeNX>**—The Ubuntu community documentation for FreeNX, a system that allows you to access your desktop from another machine over the Internet
- ▶ **www.nomachine.com**—An alternative to VNC, called FreeNX, that works much faster than VNC

CHAPTER 20

Securing Your Machines

No computer with a connection to the Internet is 100 percent safe. If this information does not concern you, it should. Although there is no way to guarantee the ability to stop patient, creative, and serious crackers who are intent on getting into your computer or network, there are ways to make it harder for them and to warn you when they do. This chapter discusses all aspects of securing your Linux machines. You might have wondered why we did not spread this information around the book wherever it is appropriate, and the reason is simple: If you ever have a security problem with Linux, you know you can turn to this page and start reading without having to search or try to remember where you saw a tip. Everything you need is here in this one chapter, and we strongly advise you to read it from start to finish.

BUILT-IN PROTECTION IN THE KERNEL

A number of networking and low-level protective services are built in to the Linux kernel. These services can be enabled, disabled, or displayed by using the `sysctl` command or by echoing a value (usually a 1 or a 0 to turn a service on or off) to a kernel process file under the `/proc` directory. See the discussion of setting a variable with `echo` in Chapter 11, “Command-Line Master Class, Part 1,” or the man pages for `sysctl` and `echo` for more.

Understanding Computer Attacks

There are many ways to classify computer attacks. Perhaps the easiest way is to characterize attacks as *internal*, or computer attacks done by someone with access to a computer on the local network, and *external*, or attacks by someone with access to a computer through the Internet.

IN THIS CHAPTER

- ▶ Understanding Computer Attacks
- ▶ Assessing Your Vulnerability
- ▶ Protecting Your Machine
- ▶ Viruses
- ▶ Configuring Your Firewall
- ▶ AppArmor
- ▶ Forming a Disaster Recovery Plan
- ▶ References

This might sound like a trivial separation to make, but it is actually important: Unless you routinely hire talented computer crackers or allow visitors to plug computers into your network, the worst internal attack you are likely to encounter is from a disgruntled employee.

HACKER VERSUS CRACKER

In earlier days, a distinction was made between the words *hacker* and *cracker*. A *hacker* was someone who used technology to innovate in new or unusual ways, whereas a *cracker* was someone who used technology to attack another's computers and cause harm. So, by the original definitions, hackers did good or cool things, and crackers did bad things.

This distinction was lost on the general public, and the term *hacker* has now regretfully come to mean the same thing as *cracker* for most people. However, we recognize the distinction and use the term *cracker* to mean a malicious person using a computer to cause problems for others. In your real-world conversations, realize that most people do not make a distinction, and so be prepared to define your terms if you call yourself a hacker.

Although you should never ignore internal threats, you should arguably be more concerned with the outside world. The big bad Internet is a security vortex. Machines connected directly to the outside world can be attacked by people around the world, and invariably are, sometimes even only a few minutes after getting connected.

This situation is not a result of malicious users lying in wait for your IP address to do something interesting. Instead, canny virus writers have created worms that exploit vulnerability, take control of a machine, and then spread to other machines around them. As a result, most attacks today result from these autocracking tools; only a handful of true clever crackers are around, and, to be frank, if one of them ever actually targets you seriously, it will take a mammoth effort to repel the cracker, regardless of what operating system you run.

Autocracking scripts also come in another flavor: prewritten code that exploits vulnerability and gives its users special privileges on the hacked machine. These scripts are rarely used by their creators; instead, they are posted online and downloaded by wannabe hackers, who then use them to attack vulnerable machines.

So, the external category is itself made up of worms, serious day job crackers, and wannabe crackers (usually called *script kiddies*). Combined, they will assault your Internet-facing servers, and your job is to make sure your boxes stay up, happily ignoring the fire-fight around them.

On the internal front, things are somewhat more difficult. Users who sit inside your firewall are already past your primary source of defense and, worse, might even have physical access to your machines. Those with malicious intent and physical access to a machine are nearly impossible to stop unless they are simply inept. The situation is only slightly better if they don't have physical access but do have access to your internal network.

Regardless of the source of the attack, you can follow a five-step checklist to help secure your box:

1. Assess your vulnerability. Decide which machines can be attacked, which services they are running, and who has access to them.
2. Configure the server for maximum security. Install only what you need, run only what you must, and configure a local firewall.
3. Secure physical access to the server.
4. Create worst-case-scenario policies.
5. Keep up to date with security news.

These steps are covered in the following sections, and all of them are equally important.

Assessing Your Vulnerability

A common mistake people make is to assume that switching on a firewall makes them safe. This is not the case and, in fact, has never been the case. Each system has distinct security needs, and taking the time to customize its security layout will give you the maximum security and the best performance.

The following list summarizes the most common mistakes:

- ▶ **Installing every package**—Do you plan to use the machine as a DNS server? If not, why have BIND installed? Go through and ensure that you have only the software you need.
- ▶ **Enabling unused services**—Do you want to administer the machine remotely? Do you want people to upload files? If not, turn off SSH and FTP because they just add needless attack vectors. Do the same for all other unused services.
- ▶ **Disabling the local firewall on the grounds that you already have a firewall at the perimeter**—In security, depth is crucial: The more layers someone has to fight through, the greater the likelihood the cracker will give up or get caught.
- ▶ **Letting your machine give out more information than it needs to**—Many machines are configured to give out software names and version numbers by default, which gives crackers a helping hand.
- ▶ **Placing your server in an unlocked room**—If you do, you might as well just turn it off now and save the worry. Even if all the employees at your company are happy and trustworthy, why take the risk?
- ▶ **Plugging your machine into a wireless network**—Unless you need wireless, avoid it, particularly if your machine is a server. Never plug a server into a wireless network because doing so is just too fraught with security problems.

After you have ruled out these potential issues, you are on to the real problem: Which attack vectors are open on your server? In Internet terms, this comes down to which services are Internet-facing and which ports they are running on.

Nmap scans your machine and reports on any open TCP/IP ports it finds. Any service you have installed that responds to Nmap's query is pointed out, which enables you to ensure that you have locked everything down as much as possible.

Nmap is available to install from the Ubuntu software repositories. Although you can use Nmap from a command line, it is easier to use with the front end—at least until you become proficient. To run the front end, open a terminal and run `nmapfe`. If you want to enable all Nmap's options, you must have administrator privileges and run `sudo nmapfe`.

When you run Nmap (by clicking the Scan button), it tests each port on your machine and checks whether it responds. If a port does respond, Nmap queries it for version information and then prints its results onscreen. The output lists the port numbers, service name (what usually occupies that port), and version number for every open port on your system. Hopefully, the information Nmap shows you will not be a surprise. If there is something open that you do not recognize, it could be that a cracker placed a back door on your system to allow easy access, and you should check into it further.

Use the output from Nmap to help find and eliminate unwanted services. The fewer services that are open to the outside world, the more secure you are. Only use Nmap on systems that you own. Scanning other people's servers is impolite, and you may also be accused of doing so in preparation for illegal activity.

Protecting Your Machine

After you have disabled all the unneeded services on your system, what remains is a core set of connections and programs that you want to keep. However, you are not finished yet: You need to clamp down your wireless network, lock your server physically, and put in place scanning procedures (such as Tripwire and promiscuous mode network monitors).

Securing a Wireless Network

Wireless networking has some unique security issues, and those issues deserve a separate discussion.

Wireless networking, although convenient, can be very insecure by its very nature because transmitted data (even encrypted data) can be received by remote devices. Those devices could be in the same room; in the house, apartment, or building next door; or even several blocks away. You must use extra care to protect the frequency used by your network. Great progress has been made in the past couple of years, but the possibility of a security breach is increased when the attacker is in the area and knows the frequency on which to listen. It should also be noted that the encryption method used by many wireless NICs is weaker than other forms of encryption (such as SSH), and you should not consider using wireless NIC encryption alone as part of your security plan.

TIP

Always use OpenSSH-related tools, such as `ssh` or `sftp`, to conduct business on your wireless LAN. Passwords are not transmitted as plain text, and your sessions are encrypted. See Chapter 19, “Remote Access with SSH and VNC,” to see how to connect to remote systems using `ssh`.

The better the physical security around your network, the more secure the network will be. (This applies to wired networks as well.) Keep wireless transmitters (routers, switches, and so on) as close to the center of your building as possible. Note or monitor the range of transmitted signals to determine whether your network is open to mobile network sniffing—a geek sport known as *war driving*. Wireshark is an example of a program that is useful for analyzing wireless traffic (as well as all other network activity). An occasional walk around your building not only gives you a break from sitting at your desk but can give you a chance to notice any people or equipment that should not be in the area.

Keep in mind that it takes only a single rogue wireless access point hooked up to a legitimate network hub to open access to your entire system. These access points can be smaller than a pack of cigarettes, and the best way to spot them is to scan for them with another wireless device.

Passwords and Physical Security

The next step toward better security is to use secure passwords on your network and ensure that users use them as well. For somewhat more physical security, you can force the use of a password with the GRUB bootloader, remove bootable devices such as DVD-ROM drives or USB boot options from the BIOS, or configure a network-booting server for Ubuntu.

Also keep in mind that some studies show that as many as 90 percent of network break-ins are done by current or former employees. If a person no longer requires access to your network, lock out access or, even better, remove the account immediately. A good security policy also dictates that any data associated with the account first be backed up and retained for a set period of time to protect against loss of important data. If you are able to do so, remove the terminated employee from the system before the employee leaves the building.

Finally, be aware of physical security. If a potential attacker can get physical access to your system, getting full access becomes trivial. Keep all servers in a locked room and ensure that only authorized personnel are given access to clients. Laptops and other mobile devices should be configured with only what is needed, and any truly sensitive data should be kept on machines that are physically secured. When machines containing any sensitive data must be used outside a secure environment, you should use hard drive encryption, such as that available when installing Ubuntu.

Something that you may find useful for laptops and other mobile devices is Prey. Prey has a basic version that is open source, free software that will help you track down a missing

device, provided that Prey was installed on the device before it was lost. There is also a paid version with a few more features. You can learn about Prey at <https://preyproject.com>.

Configuring and Using Tripwire

Tripwire is a security tool that checks the integrity of normal system binaries and reports any changes to syslog or by email. Tripwire is a good tool for ensuring that your binaries have not been replaced by Trojan horse programs. *Trojan horses* are malicious programs inadvertently installed because of identical filenames to distributed (expected) programs, and they can wreak havoc on a breached system.

There are two versions of Tripwire: an open source version and a commercial product. The free version of Tripwire is available in the Ubuntu repositories. You can find out about the differences at www.tripwire.org.

To initialize Tripwire, use its `-init` option, like this:

```
matthew@seymour~:$ sudo tripwire -init
Please enter your local passphrase:
Parsing policy file: /etc/tripwire/tw.pol
Generating the database...
*** Processing Unix File System ***
....
Wrote database file: /var/lib/tripwire/shuttle2.twd
The database was successfully generated.
```

Note that not all the output is shown here. After Tripwire has created its database (which is a snapshot of your file system), it uses this baseline along with the encrypted configuration and policy settings in the `/etc/tripwire` directory to monitor the status of your system. You should then start Tripwire in its integrity checking mode, using a desired option. (See the `tripwire` man page for details.) For example, you can have Tripwire check your system and then generate a report at the command line by entering the following:

```
matthew@seymour~:$ sudo tripwire -m c
```

No output is shown here, but you would actually get a report after entering this command. The output could be redirected to a file, but a report is saved as `/var/lib/tripwire/report/hostname-YYYYMMDD-HHMMSS.twr` (in other words, using your host's name, the year, the month, the day, the hour, the minute, and the seconds). This report can be read using the `twprint` utility, like this:

```
matthew@seymour~:$ sudo twprint --print-report -r \
/var/lib/tripwire/report/shuttle2-20020919-181049.twr | less
```

Other options, such as emailing the report, are supported by Tripwire, which should be run as a scheduled task by your system's scheduling table, `/etc/crontab`, on off-hours. (It can be resource intensive on less powerful computers.) The Tripwire software package also

includes a `twadmin` utility that you can use to fine-tune or change settings or policies or to perform other administrative duties.

Plan to spend some time reading documentation if you want to use Tripwire. It is powerful but not simple. We recommend starting with the man pages and www.tripwire.com.

Securing Devices

Do not ever advertise that you have set a NIC (network interface controller) to promiscuous mode. Promiscuous mode (which can be set on an interface by using `ifconfig`'s `promisc` option) is good for monitoring traffic across the network and can often enable you to monitor the actions of someone who might have broken into your network. The `tcpdump` command also sets a designated interface to promiscuous mode while the program runs; unfortunately, the `ifconfig` command does not report this fact while `tcpdump` is running!

Remember to use the right tool for the right job. Although you can use a network bridge to connect your network to the Internet, it would not be a good option. Bridges have almost become obsolete because they forward any packet that comes their way, which is not good when a bridge is connected to the Internet. A router enables you to filter which packets are relayed.

Viruses

In the right hands, Linux is every bit as vulnerable to viruses as Windows. This might come as a surprise to you, particularly if you made the switch to Linux on the basis of its security record. However, the difference between Windows and Linux is that Linux is much easier to secure against viruses. Indeed, as long as you are smart and diligent, you need never worry about them. Here is why:

- ▶ Linux never puts the current directory in your executable path, so typing `ls` runs `/bin/ls` rather than any program named `ls` in the current directory.
- ▶ A nonroot user can infect only the files that user has write access to, which is usually only the files in the user's home directory. This is one of the most important reasons for never using `sudo` when you don't need to.
- ▶ Linux forces you to manually mark files as executable, so you can't accidentally run a file called `myfile.txt.exe` thinking it is just a text file.
- ▶ By having more than one common web browser and email client, Linux has strength through diversity: Virus writers cannot target one platform and hit 90 percent of the users.

Despite all these factors, Linux is susceptible to being a carrier for viruses. If you run a mail server, your Linux box can send virus-infected mails on to Windows boxes. The Linux-based server would be fine, but the Windows client would be taken down by the virus. To prevent such a situation, consider using a virus scanner for your machine. You have several to choose from, both free and commercial. The most popular free suite is ClamAV (www.clamav.net), but several others compete to provide commercial solutions. Look around for the best deal before you commit.

Configuring Your Firewall

Use a hardware-based or software-based firewall on each computer connected to the Internet. Ubuntu has a firewall application named *Uncomplicated Firewall (UFW)* that is installed by default. This tool enables you to implement selective or restrictive policies regarding access to your computer or LAN.

UFW is run from the terminal, and you must have administrative privileges to use it. Commands are given like this:

```
matthew@seymour~:$ sudo ufw status
```

The most useful commands UFW are listed in Table 20.1. For others, see the `ufw` man page. Many are described in greater detail after the table.

By default, the UFW or firewall is disabled. To enable the firewall, run the following command:

```
matthew@seymour~:$ sudo ufw enable
```

To disable UFW, replace `enable` with `disable`.

Table 20.1 Useful Commands for UFW

Command	Actions Performed
Usage: <code>sudo ufw command</code>	
<code>Enable</code>	Enables the firewall
<code>Disable</code>	Disables the firewall
<code>Reload</code>	Reloads the firewall to ensure changes are applied
<code>default allow deny reject</code>	Sets default policy
<code>logging on off</code>	Toggles logging (can also be used to set the level of logging; see the man page)
<code>allow ARGS</code>	Adds an allow rule
<code>deny ARGS</code>	Adds a deny rule
<code>reject ARGS</code>	Adds a reject rule
<code>limit ARGS</code>	Adds a limit rule
<code>delete RULE</code>	Deletes the rule
<code>status shows</code>	Shows the firewall status
<code>status numbered</code>	Shows the firewall status as a numbered list of rules
Usage: <code>ufw command</code>	
<code>status verbose</code>	Shows verbose firewall status
<code>show REPORT</code>	Shows a firewall report
<code>-version</code>	Displays version information

Next, you want to enable firewall logging. Much as with the `enable` command, you run the following command:

```
matthew@seymour~:$ sudo ufw logging on
```

To enable specific ports on the firewall, you can run the `ufw` command along with the port number to open. For example, if you want to allow port 80 (HTTP) incoming connections to your Ubuntu server, enter the following:

```
matthew@seymour~:$ sudo ufw allow 80
```

To remove the firewall rule allowing port 80 connections, run the following command:

```
matthew@seymour~:$ sudo ufw delete allow 80
```

Many services are already defined in `ufw`. This means you don't have to remember the standard ports those services use, and you can allow, deny, or delete by using the service name, like this:

```
matthew@seymour~:$ sudo ufw allow ssh
```

You can also allow incoming connections from particular IP addresses. For example, if you want to let 192.168.0.1 connect to your server, enter the following:

```
matthew@seymour~:$ sudo ufw allow from 192.168.0.1
```

To remove the firewall rule allowing the previous IP address to connect, run the following command:

```
matthew@seymour~:$ sudo ufw delete allow from 192.168.0.1
```

There is a graphical interface called GFW that you can install from the Ubuntu repositories to manage UFW. The same details apply, but the interface is easier and does not require you to remember as much as with UFW.

UFW is based on `iptables`, which is used to configure the Linux kernel's built-in firewall. UFW simplifies the user tasks and syntax but really just uses `iptables`. You probably won't need to know this on your Ubuntu machine, but for completeness and to help you in case you use a different Linux distribution in the future, Table 20.2 lists simple, basic commands as a brief introduction to `iptables`.

In the table, you see words in all-caps like `ACCEPT` and `DROP`, which are policies to be set for things called chains. *Chains* are descriptions of specific types of network access, such as `INPUT`, `FORWARD`, and `OUTPUT` in Table 20.2, as well as other interactions in a network. You can, for example, define the default policy as `DROP`, which means to reject, and then `ALLOW` specific types of traffic to go through the firewall. `iptables` is quite complex and incredibly powerful.

Table 20.2 Useful Commands for `iptables`

Command	Actions Performed
Usage: <i>command</i>	
<code>sudo iptables -L</code>	Lists the currently set firewall rules
<code>sudo iptables -L -vn</code>	Lists the currently set firewall rules but with more detail
<code>sudo iptables -F</code>	Deletes all currently set firewall rules
<code>sudo iptables -P INPUT DROP</code>	Drops all incoming traffic
<code>sudo iptables -P FORWARD ACCEPT</code>	Accepts all forwarded traffic
<code>sudo iptables -P OUTPUT DROP</code>	Drops all outgoing traffic
<code>sudo iptables -A INPUT -s 8.8.8.8 -j DROP</code>	Drops all traffic from a specific IP address (the example here is a Google DNS server, and you probably don't want to block that one...)

You can block or allow traffic based on IP address, NIC, port, network, and more. You can set `iptables` to log all actions or just specific actions. You can even use it to configure NAT routers. For more, see the man pages for `iptables` and its IPv6 cousin, `ip6tables`.

`nftables` is a new firewalling tool that is gradually replacing `iptables`. `nftables` is considered stable and suitable for production environments, but it is not yet the default. If you want to start reading and consider switching now instead of waiting for it to become the default, see the official documentation at https://wiki.nftables.org/wiki-nftables/index.php/Main_Page.

AppArmor

AppArmor is a *mandatory access control (MAC)* system. It is less complicated than the better-known SELinux (<https://selinuxproject.org/>), a MAC framework created by the U.S. National Security Agency (NSA). AppArmor is designed to limit what specific programs can do by restricting them to the use of predetermined resources—and only those resources. This is done via profiles, which are loaded into the kernel at boot. It can be run in complain mode, where information is logged about insecure practices but no action is taken, or in enforce mode, where policies and limits are active.

This section provides a brief introduction to AppArmor. For a fuller introduction, check the links listed in the “References” section at the end of this chapter.

By default, AppArmor does little. You can install some extra profiles from the Ubuntu repositories by installing the `apparmor-profiles` package. These run in complain mode and log issues in `/var/log/syslog`.

To unleash the power of AppArmor, you need to edit or create text files in `/etc/apparmor.d`. A profile is named for the application it restricts, including the full path to the application in the file system. For example, the file `sbin.rsyslogd`, shown here, restricts the system logging daemon:

```
# Last Modified: Sun Sep 25 08:58:35 2011
```

```
#include <tunables/global>
# Debugging the sysloger can be difficult if it can't write to the file# that the ker-
nel is logging denials to. In these cases, you can do the# following:# watch -n 1 'dmesg
| tail -5'
/usr/sbin/rsyslogd { #include <abstractions/base> #include <abstractions/nameservice>
    capability sys_tty_config, capability dac_override, capability dac_read_search,
    capability setuid, capability setgid, capability sys_nice, capability syslog,
    # rsyslog configuration /etc/rsyslog.conf r, /etc/rsyslog.d/ r, /etc/rsyslog.d/** r,
    /{/var}/run/rsyslogd.pid rwk, /var/spool/rsyslog/ r, /var/spool/rsyslog/** rwk,
    /usr/lib{,32,64}/rsyslog/*.so mr,
    /dev/tty* rw, /dev/xconsole rw, @{/PROC}/kmsg
r,
    /dev/log wl, /var/lib*/dev/log wl, /var/spool/post-
fix/dev/log wl,
    # 'r' is needed when using imfile /var/log/** rw,
    # Add these for mysql support #/etc/mysql/my.cnf r, #/{/var}/run/mysqld/mysqld.sock rw,
    # Add these for postgresql support ##include <abstractions/openssl> ##include
<abstractions/ssl_certs> #/{/var}/run/postgresql/s.PGSQL.*[0-9] rw,
    # Site-specific additions and overrides. See local/README for details. #include
<local/usr.sbin.rsyslogd>}
```

Even without knowing the syntax, you can see from this file that profiles are simple text files that support the use of comments, that absolute paths and globbing (pattern matching for filenames) are supported, that specific capabilities are allowed or disallowed, and what locations and programs in the file system may be accessed or used.

Each access rule specifies specific permissions from this list:

```
r - read
w - write
ux - unconstrained execute
Ux - unconstrained execute - scrub the environment
px - discrete profile execute
Px - discrete profile execute - scrub the environment
ix - inherit execute
m - allow PROT_EXEC with mmap(2) calls
l - link
```

These permissions are listed at the ends of lines.

genprof is a program that helps you generate or update a profile. You supply the name of the executable (or the path, if it is not already in the path) and may optionally supply the path to the profiles, as well. The following is an example:

```
matthew@seymour~:$ sudo genprof google-chrome
```

You will be asked to start the program and use it for a bit. After it is complete, you are given an opportunity to choose whether access to each function should be allowed

or denied. The program then writes a text file in `/etc/apparmor.d`, using the name of the program and its path (in this case, `opt.google.chrome.google-chrome`, which in this case was installed directly from Google (www.google.com/chrome?platform=linux) so no premade AppArmor profile exists on the system used in this example). You may then edit the text file as desired, which you must do if you want to change from complain mode to enforce mode.

When you have a set of profiles that cover what you need, these are the commands you will use most often:

- ▶ **start:** Use the `start` command as follows:

```
matthew@seymour~:$ sudo service apparmor start
```

- ▶ **stop:** Use the `stop` command as follows:

```
matthew@seymour~:$ sudo service apparmor stop
```

- ▶ **reload:** Use the `reload` command as follows:

```
matthew@seymour~:$ sudo service apparmor reload (or restart)
```

- ▶ **show status:** Use the `show status` command as follows:

```
matthew@seymour~:$ sudo service apparmor status
```

This section has just scratched the surface of AppArmor, but hopefully you have learned enough information that your appetite has been whetted, and you are ready to do some further reading.

Forming a Disaster Recovery Plan

No one likes planning for the worst, which probably explains why two-thirds of people do not have wills. Having your systems hacked is a scary thing: One or more criminals has broken through your carefully laid blocks and caused untold damage to the machine. Your boss, if you have one, wants a full report of what happened and why, and your users want their email when they sit down at their desks in the morning. What do you do?

If you ever do get hacked, nothing will take the stress away entirely. However, if you take the time to prepare a proper response in advance, you should at least avoid premature aging. Here are some suggestions to get you started:

- ▶ **Do not just pull out the network cable.** Pulling the cable acts as an alert that the cracker has been detected, which rules out any opportunities for security experts to monitor for that cracker returning and actually catch him or her.
- ▶ **Inform only the people who need to know.** Your boss and other IT people are at the top of the list; other employees are not. Keep in mind that it could be one of the employees behind the attack, and you don't want to tip off the culprit.

- ▶ **If the machine is not required, and you do not want to trace the attack, you can safely remove it from the network.** However, do not switch it off because some backdoors are enabled only when the system is rebooted.
- ▶ **Make a copy of all the log files on the system and store them somewhere else.** These files might have been tampered with, but they might contain nuggets of information.
- ▶ **Check the `/etc/passwd` file and look for users you do not recognize.** Change all the passwords on the system and remove bad users.
- ▶ **Check the output of `ps aux` to see if unusual programs are running.** Also check to see whether any `cron` jobs are set to run.
- ▶ **Look in `/var/www` and see whether any web pages are there that should not be.**
- ▶ **Check the contents of the `.bash_history` files in the `/home` directories of your users.** Are there any recent commands for your primary user?
- ▶ **If you have worked with external security companies previously, call them in for a fresh audit.** Hand over all the logs you have and explain the situation. They will be able to extract all the information from the logs that is possible to extract.
- ▶ **Start collating backup tapes from previous weeks and months.** Your system might have been hacked long before you noticed, and you might need to roll back the system more than once to find out when the attack actually succeeded.
- ▶ **Download and install Rootkit Hunter from <http://rkhunter.sourceforge.net/>.** This tool searches for and removes the types of files that bad guys leave behind for their return.

Finally, keep your disaster recovery plan somewhere safe; saving it as a file on the machine in question is a *very* bad move.

References

- ▶ <https://help.ubuntu.com/community/InstallingSecurityTools>—Ubuntu community documentation of and suggestions for various security tools
- ▶ <https://nmap.org>—The official site for Nmap
- ▶ www.tripwire.org—Information and download links for the open source version of Tripwire
- ▶ www.ubuntu.com/usn—The official Ubuntu security notices list, which is well worth watching

- ▶ <https://help.ubuntu.com/community/UFW>—Ubuntu community documentation for UFW
- ▶ <https://wiki.ubuntu.com/UncomplicatedFirewall>—Ubuntu documentation for UFW
- ▶ <https://gitlab.com/apparmor/apparmor/wikis/home/>—The official documentation for AppArmor
- ▶ <https://help.ubuntu.com/community/AppArmor>—Ubuntu community documentation for AppArmor
- ▶ <https://wiki.ubuntu.com/Security/Features>—An Ubuntu wiki page that outlines and describes security features, Ubuntu, and configuration

CHAPTER 21

Performance Tuning

Squeezing extra performance out of your hardware might sound like a pointless task, given the low price of commodity upgrades today. To a certain degree, that is true; for most of us, it is cheaper to buy a new computer than to spend hours fighting to get a 5 percent speed boost. But what if the speed boost were 20 percent? How about if it were 50 percent?

The amount of benefit you can get by optimizing your system varies depending on what kinds of tasks you are running, but there is something for everyone. Over the next few pages we look at quick ways to optimize the Apache web server, both the KDE and GNOME desktop systems, both MySQL and PostgreSQL database servers, and more.

Before we get into the details, you need to understand that *optimization* is not an absolute term: If you optimize a system, you have improved its performance, but it is still possible it could further be increased. You are typically not interested in getting 99.999 percent performance out of a system because optimization suffers from the law of diminishing returns: Some basic changes make the biggest differences, but after that, it takes increasing amounts of work to obtain decreasing speed improvements.

Many Linux users love to tinker under the hood to increase the performance of their computers, and Linux gives you some great tools to do that. Whereas stability-loving nurturers generally tell us, “Don’t fix what’s not broken,” experiment-loving hot-rodders often say, “Fix it until it breaks.” In this section, you learn about many of the commands used to tune, or “tweak,” your file system.

Before you undertake any under-the-hood work with Linux, however, keep a few points in mind. First, perform a benchmark on your system before you begin. Linux does not

IN THIS CHAPTER

- ▶ Storage Disk
- ▶ Kernel
- ▶ Tuned
- ▶ References

offer a well-developed benchmarking application, and availability of what exists changes rapidly. You can search online for the most up-to-date information for benchmarking applications for Linux. If you are a system administrator, you might choose to create your own benchmarking tests. Second, remember to tweak only one thing at a time so you can tell what works, what does not work, and what breaks things. Some of these tweaks might not work or might lock up your machine, but if you are only implementing them one at a time, you will find it much easier to reverse a change that caused a problem.

If you aren't measuring, you aren't tuning. Measure what you want to improve before making changes, as you make incremental changes, and after you think you are done making changes. Don't assume anything, ever. Only then can you be sure what you have done was/is useful.

Always have a working boot disc handy, such as the live Ubuntu CD or DVD. Remember that you are personally assuming all risks for attempting any of these tweaks. If you don't understand what you are doing or are not confident in your ability to revert any changes discussed here, do not attempt any of the suggestions in this chapter. The default settings in Ubuntu work very well for most people and really don't need adjusting—just as most people can use and enjoy their car just as it is. However, some people love taking apart their cars and building hot-rods; they enjoy tweaking and breaking and fixing them. This chapter is for that sort of person. If you don't think you can fix it, don't risk breaking it.

Storage Disk

This first set of tuning ideas all have to do with accessing and storing your data. There are often tradeoffs involved when tuning for performance, such as reducing or eliminating redundancy or journaling. For some data that is ephemeral and trivial if lost, that's fine. For most data, you want to keep all of these intact to help ensure data persistence and integrity.

Also, with solid state drives and the speed increase seen when using them versus using hard drives, there really isn't the compelling need to tune most of these things anymore.

Linux File Systems

Most of the options discussed in this chapter are for the `ext2`, `ext3`, or `ext4` file systems. The default in Ubuntu—and most other current Linux distributions today—is `ext4`. It is also common to see `xfs` and `zfs` systems.

Both `ext3` and `ext4` are journaling file systems, which are much more reliable than other file systems for data storage. A journaling file system writes data changes first to a log in a separate part of the drive. Once journaled, the change is queued and performed in the background. This can help prevent data corruption while also keeping data in a consistent state. If there is a problem, the data can be rebuilt from the journal.

Other file system options are available. Most of them are considered legacy options or are used for compatibility with other operating systems, such as NTFS, used by Microsoft Windows.

The file system receiving the most attention currently is ZFS. ZFS comes from Sun Microsystems and the Solaris operating system (now owned by Oracle) and has been long loved in data centers. However, ZFS is proprietary and was unavailable for other operating systems—but not anymore. Well, it is available, but it is still proprietary licensed, so there is a potential risk involved for anything but personal use. This option is not recommended for casual use, but if you have large-scale storage needs and are willing to do a bit of reading and configuration, you may find it incredibly useful.

The `hdparm` Command

The `hdparm` utility can be used by root to set and tune the settings for IDE hard drives. You would do this to tune the drives for optimal performance. After previously requiring a kernel patch and installation of associated support programs, the `hdparm` program is now included with Ubuntu. You should only experiment with the file systems mounted read-only because some settings can damage some file systems when used improperly.

The general format of the command is this:

```
matthew@seymour:~$ hdparm command device
```

The following command runs a hard disk test:

```
matthew@seymour:~$ hdparm -tT /dev/hda
```

You must replace `/dev/hda` with the location of your hard disk. `hdparm` then runs two tests: cached reads and buffered disk reads. A good IDE hard disk should be getting 400Mbps to 500Mbps for the first test and 20Mbps to 30Mbps for the second. Note your scores and then try this command:

```
matthew@seymour:~$ hdparm -m16 -d1 -u1 -c1 /dev/hda
```

This enables various performance-enhancing settings. Now try executing the original command again. If your scores increase from the previous measurement, you should run this command:

```
matthew@seymour:~$ hdparm -m16 -d1 -u1 -c1 -k1 /dev/hda
```

The extra parameter tells `hdparm` to write the settings to disk so that they will be used each time you boot, ensuring optimal disk performance in the future.

The man page entry for `hdparm` is extensive and contains useful detailed information, but because the kernel configuration selected by Ubuntu already attempts to optimize the drives, it might be that little can be gained through tweaking. Because not all hardware combinations can be anticipated by Ubuntu or by Linux, and performance gains are always useful, you're encouraged to try.

TIP

You can use the `hdparm` command to produce a disk transfer speed result as follows:

```
matthew@seymour:~$ hdparm -tT device
```

Be aware, however, that although the resulting numbers appear quantitative, they are subject to several technical qualifications beyond the scope of what is discussed and explained in this chapter. Simply put, do not accept values generated by `hdparm` as absolute numbers but only as a relative measure of performance.

File System Tuning

Never content to leave things alone, Linux provides several tools to adjust and customize file system settings. The belief is that hardware manufacturers and distribution creators tend to select conservative settings that will work well all the time, leaving some of the potential of your system leashed—which is why you have chosen *Ubuntu Unleashed* to help you.

The Linux file system designers have done an excellent job of selecting default values used for file system creation, and any version of the Linux kernel after 2.6.x contains code for the IDE subsystem that significantly improves *I/O (input/output)* transfer speeds over older versions, obviating much of the need for special tweaking of the file system and drive parameters if you use IDE disks. Although these values work well for most users, some server applications of Linux benefit from file system tuning. As always, observe and benchmark your changes.

SYNCHRONIZING THE FILE SYSTEM WITH `sync`

Because Linux uses buffers when writing to devices, a write will not occur until the buffer is full, until the kernel tells it to, or if you tell it to by using the `sync` command. Traditionally, the command is given twice, as in the following example:

```
matthew@seymour:~$ sync ; sync
```

To do it twice is overkill. Still, it can be helpful before the unmounting of certain types of media with slow write speeds (such as some USB hard drives or PCMCIA storage media)—but only because it delays the user from attempting to remove the media too soon rather than because two syncs are better than one.

The `tune2fs` Command

With `tune2fs`, you can adjust the tunable file system parameters on an `ext2`, `ext3`, or `ext4` file system. A few performance-related items of note are as follows:

- ▶ To disable file system checking, the `-c 0` option sets the maximal mount count to zero.
- ▶ The interval between forced checks can be adjusted with the `-i` option.

- ▶ The `-m` option sets the reserved blocks percentage with a lower value, freeing more space at the expense of `fsck` having less space to write any recovered files.
- ▶ Decrease the number of superblocks to save space with the `-O sparse_super` option. (Modern file systems use this by default.) Always run `e2fsck` after you change this value.
- ▶ More space can be freed with the `-r` option, which sets the number of reserved (for root) blocks.

Note that most of these uses of `tune2fs` free up space on the drive at the expense of the capability of `fsck` to recover data. Unless you really need the space and can deal with the consequences, just accept the defaults; large drives are now relatively inexpensive.

The `e2fsck` Command

The `e2fsck` utility checks an `ext2/ext3/ext4` file system. Some useful arguments taken from `man e2fsck` are as follows:

- ▶ `-c`—Checks for bad blocks and then marks them as bad
- ▶ `-f`—Forces checking on a clean file system
- ▶ `-v`—Enters verbose mode

The `badblocks` Command

Although not a performance-tuning program per se, the utility `badblocks` checks a partition (preferably unmounted) for bad blocks. Running this command by itself is not recommended; rather, you should allow it to be called by `fsck`. You should use it directly only if you specify the block size accurately; don't guess or assume anything.

The options available for `badblocks` are detailed in the man page. They allow for very low-level manipulation of the file system that is useful for data recovery by file system experts or for file system hacking, but they are beyond the scope of this chapter and the average user.

Disabling File Access Time

Whenever Linux reads a file, it changes the last access time (known as the *atime*). This is also true for your web server: If you are getting hit by 50 requests a second, your hard disk will be updating the *atime* 50 times a second. Do you really need to know the last time a file was accessed? If not, you can disable the *atime* setting for a directory by typing this:

```
matthew@seymour:~$ chattr -R +A /path/to/directory
```

The `chattr` command changes file system attributes, of which “don't update *atime*” is one. To set that attribute, use `+A` and specify `-R` so that it is recursively set. `/path/to/directory` gets changed, and so do all the files and subdirectories it contains.

Kernel

As the Linux kernel developed over time, developers sought a way to fine-tune some of the kernel parameters. Before `sysctl`, those parameters had to be changed in the kernel configuration, and then the kernel had to be recompiled.

The `sysctl` command can change some parameters of a running kernel. It does this through the `/proc` file system, which is a “virtual window” into the running kernel. Although it might appear that a group of directories and files exist under `/proc`, that is only a representation of parts of the kernel. When you’re the root user (or using the `sudo` command), you can read values from and write values to those “files,” referred to as *variables*. You can display a list of the variables as shown in the following example (which presents an abbreviated list because roughly 250 items or more exist in the full list):

```
matthew@seymour:~$ sysctl -A
net.ipv4.tcp_max_syn_backlog = 1024
net.ipv4.tcp_rfc1337 = 0
net.ipv4.tcp_stdurg = 0
net.ipv4.tcp_abort_on_overflow = 0
net.ipv4.tcp_tw_recycle = 0
net.ipv4.tcp_syncookies = 0
net.ipv4.tcp_fin_timeout = 60
net.ipv4.tcp_retries2 = 15
net.ipv4.tcp_retries1 = 3
net.ipv4.tcp_keepalive_intvl = 75
net.ipv4.tcp_keepalive_probes = 9
net.ipv4.tcp_keepalive_time = 7200
net.ipv4.ipfrag_time = 30
```

The items shown are networking parameters, and tweaking these values is beyond the scope of this book. If you want to change a value, however, you use the `-w` parameter:

```
matthew@seymour:~$ sysctl -w net.ipv4.tcp_retries 2=20
```

This increases the value of that particular kernel parameter.

If you find that a particular setting is useful, you can enter it into the `/etc/sysctl.conf` file. The format is as follows, using the earlier example:

```
net.ipv4.tcp_retries 2=20
```

Of more interest to kernel hackers than regular users, `sysctl` is a potentially powerful tool that continues to be developed and documented.

TIP

The kernel does a good job of balancing performance for graphical systems, so there’s not a great deal you can do to tweak your desktop to run faster.

Both GNOME and KDE are “heavyweight” desktop systems: They are all-inclusive, all-singing, and all-dancing environments that do far more than browse your file system. The drawback is that their size makes them run slowly on older systems. On the flip side, Ubuntu has other systems available in the repositories, like the Xfce and LXDE desktops, which are a great deal slimmer and faster than GNOME and KDE. If you find that GNOME and KDE are struggling just to open a file browser, Xfce or LXDE are likely to work for you.

Tuned

The Tuned Project provides a system tuning service for Linux. Tuned does many things to help you optimize your system. Tuned is a profile-based system that uses the udev device manager to monitor connected devices and also enables both static and dynamic tuning of system settings according to your chosen profile. It can drastically improve performance.

Tuned does the following:

- ▶ Monitors connected devices
- ▶ Tunes system settings using a profile you designate
- ▶ Supports configurations like `sysctl`, `sysfs`, and kernel boot command-line parameters
- ▶ Support hot-plugging of devices
- ▶ Can be run in `no-daemon` mode, which reduces functionality, which is needed in secure environments
- ▶ Stores its configuration in one place
- ▶ Allows hierarchical definitions of profiles through inheritance
- ▶ Supports full rollback of changes it makes
- ▶ Includes a set of predefined profiles for common use cases while also allowing the creation of custom profiles

To begin, install Tuned.

```
$ sudo apt install tuned
```

To view a list of available profiles for you to use, enter

```
$ sudo tuned-adm list
```

Listed profiles include short descriptions.

To show the profile currently in use, enter

```
$ sudo tuned-adm active
```

To show the recommended profile for your system, enter

```
$ sudo tuned-adm recommend
```

To change the profile currently in use, enter this, where *profilename* is the name of the profile you desire to switch over to using:

```
$ sudo tuned-adm profile profilename
```

To verify that the system is currently using the settings in your current profile (checking to see whether a setting has perhaps been directly modified after the Tuned profile was set), enter

```
$ sudo tuned-adm verify
```

To unload all tunings made by Tuned, enter

```
$ sudo tuned-adm off
```

The Tuned daemon is controlled by *systemd* using standard *systemctl* commands like *start*, *stop*, and *restart*. Tuned is incredibly easy to use and in some settings can make a big difference, so it's worth checking out. For more information, see <https://tuned-project.org/>.

References

- ▶ **www.coker.com.au/bonnie++/**—The home page of Bonnie++, a disk benchmarking tool, which also contains a link to RAID benchmarking utilities and Postal, a benchmarking utility for SMTP servers
- ▶ **www.phoronix-test-suite.com**—Home of the Phoronix Test Suite, which does automated performance testing and comparisons and is a quality benchmarking software option to consider
- ▶ **<https://tuned-project.org>**—The official website for Tuned

CHAPTER 22

Kernel and Module Management

A kernel is a complex piece of software that manages the processes and process interactions that take place within an operating system. As a user, you rarely, if ever, interact directly with it. Instead, you work with the applications that the kernel manages.

The Linux kernel is Linux. It is the result of years of cooperative (and sometimes contentious) work by numerous people around the world. There is only one common kernel source tree, but each major Linux distribution massages and patches its version slightly to add features, performance, or options. Each Linux distribution, including Ubuntu, comes with its own precompiled kernel as well as the kernel source code, providing you with absolute authority over the Linux operating system. This chapter covers the kernel and what it does for you and for the operating system.

In this chapter, you also learn how to obtain the kernel sources and how and when to patch the kernel. This chapter leads you through an expert's tour of the kernel architecture and teaches you essential steps in kernel configuration, how to build and install modules, and how to compile drivers in Ubuntu. This chapter also teaches you important aspects of working with GRUB2, the default Ubuntu boot loader. Finally, the chapter's troubleshooting information will help you understand what to do when something goes wrong with your Linux kernel installation or compilation process. As disconcerting as these problems can seem, this chapter shows you some easy fixes for many kernel problems.

Almost all users find that a precompiled Ubuntu kernel suits their needs (and there are several to choose from). At some

IN THIS CHAPTER

- ▶ The Linux Kernel
- ▶ Managing Modules
- ▶ When to Recompile
- ▶ Kernel Versions
- ▶ Obtaining the Kernel Sources
- ▶ Patching the Kernel
- ▶ Compiling the Kernel
- ▶ When Something Goes Wrong
- ▶ References

point, you might need to recompile the kernel to support a specific piece of hardware or add a new feature to the operating system, although the Ubuntu kernel team works very hard to backport or enable any feature possible (as a module), so it is highly unlikely you will ever have a need to do this. The kernel team is also approachable and will gladly discuss specific needs and features. Sometimes features are not enabled just because no one has ever asked for or needed them. Occasionally, things are not enabled because of a conflict with another feature. The Ubuntu kernel team can help you discover what is going on in those cases (but don't abuse their kindness and availability; they already work quite hard and for long hours).

Really, the main reason today that people compile their own kernel is because they want to learn to be a kernel developer. If you have heard horror stories about the difficulties of recompiling the Linux kernel, you can relax; this chapter gives you all the information you need to understand how to painlessly work through the process if you are interested in learning a new skill. This is a complex and detail-oriented task, but it is within the grasp of most technical users, even if it is completely unnecessary.

CAUTION

Building and using a custom kernel will make it difficult to get support for your system. Although it is a learning experience to compile your own kernel, you will not be allowed to file bugs in Ubuntu on a custom-built kernel (if you do, they will be rejected without further explanation), and if you have a commercial support contract with Ubuntu/Canonical, building and using a custom kernel will void the contract.

The Linux Kernel

The Linux kernel is the management part of the operating system that many people call *Linux*. Although many think of the entire distribution as Linux, the only piece that can correctly be called Linux is the kernel. Ubuntu, like many other Linux distributions, includes a kernel packaged with add-on software that interacts with the kernel so that the user can interface with the system in a meaningful manner.

The system utilities and user programs enable computers to become valuable tools to a user.

THE FIRST LINUX KERNEL

In 1991, Linus Torvalds released version 0.99 of the Linux kernel as the result of his desire for a powerful, UNIX-like operating system for his Intel 80386 personal computer. Linus wrote the initial code necessary to create what is now known as the Linux kernel and combined it with Richard Stallman's GNU tools. Indeed, because many of the Linux basic system tools come from the GNU Project, many people refer to the operating system as *GNU/Linux*. Since then, Linux has benefited from thousands of contributors adding their talents and time to the Linux project. Linus still maintains the kernel, deciding what will and will not make it into the kernel as official releases, known to many as the *vanilla* or *Linus Linux* kernel.

The Linux Source Tree

The source code for the Linux kernel is kept in a group of directories called the *kernel source tree*. The structure of the kernel source tree is important because the process of compiling (building) the kernel is automated; it is controlled by scripts interpreted by the `make` application. These scripts, known as *makefiles*, expect to find the pieces of the kernel code in specific places; if they don't find them, they will not work. You learn how to use `make` to compile a kernel later in this chapter.

It is not necessary for the Linux kernel source code to be installed on your system for the system to run or for you to accomplish typical tasks such as working with email, browsing the Web, or using a word processing program. It is necessary that the kernel sources be installed, however, if you want to compile a new kernel. In the next section, you learn how to install the kernel source files and how to set up the special symbolic link required. That link, `/usr/src/linux-5.4.0`, where the numbers at the end match the release version of the kernel installed on your system, is what we use to refer to the directory of the kernel source tree as we examine the contents of the kernel source tree.

NOTE

The pace of change in the Linux kernel has accelerated, much like the rest of our lives. In this chapter, we chose to use the version numbers for a recent LTS release of Ubuntu. The version numbers on your system may be different, but the processes and concepts remain the same.

The `/usr/src/linux-5.4.0` directory contains the `.config` files and the makefiles, among others. The `.config` file is the configuration of your Linux kernel as it was compiled. There is no `.config` file by default; you must select one from the `/configs` subdirectory. There, you will find configuration files for each flavor of the kernel Ubuntu provides; simply copy the one appropriate for your system to the default directory and rename it `.config`.

We have already discussed the contents of the `/configs` subdirectory, so let's examine the other directories found under `/usr/src/linux-5.4.0`. The most useful one for us is the `Documentation` directory. In it and its subdirectories, you will find almost all the documentation concerning every part of the kernel. The file `00-INDEX` (which each `Documentation` subdirectory contains) contains a list of the files in the main directory and a brief explanation of what they are. Many files are written solely for kernel programmers and application writers, but a few are useful to intermediate or advanced Linux users when attempting to learn about kernel and device driver issues. Some of the more interesting and useful documents are as follows:

- ▶ **devices.txt**—This file contains a list of all possible Linux devices that are represented in the `/dev` directory, giving major and minor numbers and short descriptions. You may at some point have gotten an error message that mentions `char-major-xxx`; this file is where that list is kept.

- ▶ **ide.txt**—If your system uses IDE hard drives, this file discusses how the kernel interacts with them and lists the various kernel commands that you can use to solve IDE-related hardware problems, manually set data transfer modes, and otherwise manually manage your IDE drives. Most of this management is automatic, but if you want to understand how the kernel interacts with IDE devices, this file explains it.
- ▶ **initrd.txt**—This file provides much more in-depth knowledge of initial RAM disks, giving details on the loopback file system used to create and mount them and explaining how the kernel interacts with them.
- ▶ **kernel-parameters.txt**—This file is a list of most of the arguments that you can pass at boot time to configure kernel or hardware settings, but it does not appear too useful at first glance because it is just a list. However, knowing that a parameter exists and might relate to something you are looking for can assist you in tracking down more information because it gives you terms to enter into an Internet search engine such as Google.
- ▶ **sysrq.txt**—If you have ever wondered what that key on your keyboard marked SysRq is used for, this file has the answer. Briefly, it is a key combination hardwired into the kernel that can help you recover from a system lockup. Ubuntu disables this function by default for security reasons. You can re-enable it at a root prompt by entering the command `# echo "1" > /proc/sys/kernel/sysrq`, and you can disable it by echoing the value `0` rather than `1`.

In the other directories under `Documentation`, you find similar text files that deal with the kernel modules for CD-ROM drivers, file system drivers, game port and joystick drivers, video drivers (not graphics card drivers; those belong to X11R6 and not to the kernel), network drivers, and all the other drivers and systems found in the Linux operating system. Again, these documents are usually written for programmers, but they can also provide useful information to intermediate and advanced Linux users.

The directory named `scripts` contains many of the scripts that `make` uses. It really does not contain anything of interest to anyone who is not a programmer or a kernel developer (also known as a *kernel hacker*).

After a kernel is built, all the compiled files wind up in the `arch` directory and its subdirectories. Although you can manually move them to their final location, you learn later in this chapter how the `make` scripts will do it for you. In the early days of Linux, this post-compilation file relocation was all done by hand; you should be grateful for `make`.

NOTE

The `make` utility is a complex program. You can find complete documentation on the structure of `make` files, as well as the arguments that it can accept, at www.gnu.org/software/make/manual/make.html.

The remaining directories in `/usr/src/linux-5.4.0` contain the source code for the kernel and the kernel drivers. When you install the kernel sources, these files are placed

there automatically. When you patch kernel sources, these files are altered automatically. When you compile the kernel, these files are accessed automatically. Although you never need to touch the source code files, they can be useful. The kernel source files are nothing more than text files with special formatting, which means you can look at them and read the programmer comments. Sometimes, a programmer writes an application but cannot (or often does not) write the documentation. The comments the programmer puts in the source code are often the only documentation that exists for the code.

Small testing programs are even “hidden” in the comments of some of the code, along with comments and references to other information. Because the source code is written in a language that can be read as easily—almost—as English, a nonprogrammer might be able to get an idea of what the application or driver is actually doing (see Chapter 38, “Using Programming Tools,” for an idea of how that could happen). This information might be of use to an intermediate to an advanced Linux user who is confronted by kernel- and driver-related problems.

NOTE

The interaction and control of hardware is handled by a small piece of the kernel called a *device driver*. The driver tells the computer how to interact with a modem, a SCSI card, a keyboard, a mouse, and so on in response to a user prompt. Without the device driver, the kernel does not know how to interact with the associated device.

Types of Kernels

In the early days of Linux, a kernel was a single block of code containing all the instructions for the processor, the motherboard, and the other hardware. If you changed hardware, you were required to recompile the kernel code to include what you needed and discard what you did not need. Including extra, unneeded code carried a penalty because the kernel became larger and occupied more memory. On older systems that had only 4MB to 8MB of memory, wasting precious memory for unnecessary code was considered unacceptable. Kernel compiling was something of a “black art,” as early Linux users attempted to wring the most performance possible from their computers. Such kernels compiled as a single block of code are called *monolithic kernels*.

As the kernel code grew larger and the number of devices that could be added to a computer increased, the requirement to recompile became onerous. A new method of building the kernel was developed to make the task of compiling easier. The part of the kernel’s source code that composed the code for the device drivers could be optionally compiled as a module that could be loaded and unloaded into the kernel as required. This is known as the *modular* approach to building the kernel. Now, all the kernel code could be compiled at once, with most of the code compiled into these modules. Only the required modules would be loaded; the kernel could be kept smaller, and adding hardware was much simpler.

The typical Ubuntu kernel has some drivers compiled as part of the kernel itself (called *inline drivers*) and others compiled as modules. Only device drivers compiled inline are available to the kernel during the boot process; modular drivers are available only after the system has been booted.

NOTE

As a common example, drivers for SCSI disk drives must be available to the kernel if you intend to boot from SCSI disks. If the kernel is not compiled with those drivers inline, the system does not boot because it cannot access the disks.

A way around this problem for modular kernels is to use an initial RAM disk (`initrd`), discussed later in this chapter, in the “Creating an Initial RAM Disk Image” section. `initrd` loads a small kernel and the appropriate device driver, which then can access the device to load the kernel you want to run.

Some code can be only one or the other (for technical reasons unimportant to the average user), but most code can be compiled either as modular or inline. Depending on the application, some system administrators prefer one way over the other, but with fast modern processors and abundant system memory, the performance differences are of little concern to all but the most ardent Linux hackers.

When compiling a kernel, making the selection of modular or inline is part of the `make config` step, detailed later in this chapter. Unless you have a specific reason to do otherwise, select the modular option when given a choice. Because you will be managing kernels more frequently than compiling kernels, the process of managing modules is addressed in the next section.

Managing Modules

When using a modular kernel, special tools are required to manage the modules. Modules must be loaded and unloaded, and it would be nice if that were done as automatically as possible. You also need to be able to pass necessary parameters to modules when you load them—things such as memory addresses and interrupts. (That information varies from module to module, so you need to look at the documentation for your modules to determine what, if any, information needs to be passed to them.) This section covers the tools provided to manage modules and then looks at a few examples of using them.

Linux provides the following module management tools, all of which (along with `modprobe.conf`) have man pages:

- ▶ **lsmod**—This command lists the loaded modules. It is useful to pipe this through the `less` command because the listing is usually more than one page long.
- ▶ **insmod**—This command loads the specified module into the running kernel. If a module name is given without a full path, the default location for the running kernel, `/lib/modules/*/`, is searched. Several options are offered for this command; the most useful is `-f`, which forces the module to be loaded.
- ▶ **rmmod**—This command unloads (removes) the specified module from the running kernel. More than one module at a time can be specified.
- ▶ **modprobe**—A more sophisticated version of `insmod` and `rmmod`, this command uses the dependency file created by `depmod` and automatically handles loading or, with the `-r` option, removing modules. There is no force option, however. A useful option to `modprobe` is `-t`, which causes `modprobe` to cycle through a set of drivers until it

finds one that matches your system. If you are unsure of what module will work for your network card, use this command:

```
matthew@seymour:~$ sudo modprobe -t net
```

The term `net` is used here because that is the name of the directory (`/lib/modules/* /kernel/net`) where all the network drivers are kept. The `modprobe` command tries each one in turn until it loads one successfully.

- ▶ **modinfo**—This command queries a module’s object file and provides a list of the module name, author, license, and any other information that is there. It often is not very useful.
- ▶ **depmod**—This program creates a dependency file for kernel modules. Some modules need to have other modules loaded first; that is, they “depend” on the other modules. (A lot of the kernel code is like this because it eliminates redundancy in the code base.) During the boot process, one of the startup files contains the command `depmod -a`, and it is run every time you boot to re-create the file `/lib/modules/*/modules.dep`. If you make changes to the file `/etc/modprobe.conf`, run `depmod -a` manually. The `depmod` command, its list of dependencies, and the `/etc/modprobe.conf` file enable kernel modules to be automatically loaded as needed.
- ▶ **/etc/modprobed**—This directory contains a set of `*.conf` files to specify options for modules that have dependencies. They can also be used to create aliases or override the default `modprobe` behavior for modules with special requirements.
- ▶ **/etc/modprobe.conf**—This is not a command but a file that controls how `modprobe` and `depmod` behave; it contains kernel module variables. Although the command syntax can be quite complex, most actual needs are simple. The most common use is to alias a module and then pass some parameters to it. For example, the following code aliases a device name (from `devices.txt`) to a more descriptive word and then passes some information to an associated module:

```
alias char-major-89 i2c-dev
options eeprom ignore=2,0x50,2,0x51,2,0x52
```

The `i2c-dev` device is used to read the CPU temperature and fan speed on this system. These lines for `/etc/modprobe.conf` were suggested for our use by the program’s documentation. We added them with a text editor.

A partial listing of `lsmod` is shown here, piped through the `less` command, enabling you to view it a page at a time:

```
matthew@seymour:~$ sudo lsmod | less
Module          Size Used by
parport_pc      19392  1
Module          Size Used by
parport_pc      19392  1
lp              8236   0
joydev          17377   0
parport         29640  2 parport_pc,lp
autofs4         10624   0
sunrpc          101064  1
```

The list is actually much longer, but here you see that the `input` module is being used by the `joydev` (joystick device) module, but the joystick module is not being used. This computer has a joystick port that was autodetected, but no joystick is connected. A scanner module is also loaded, but because the USB scanner is unplugged, the module is not being used. You use the `lsmod` command to determine whether a module was loaded and what other modules were using it. If you examine the full list, you see modules for all the devices attached to your computer.

To remove a module, `joydev` in this example, use the following:

```
matthew@seymour:~$ sudo rmmod joydev
```

or use this:

```
matthew@seymour:~$ sudo modprobe -r joydev
```

The output of `lsmod` now shows that it is no longer loaded. If you were to remove `input` as well, you could then use `modprobe` to load both `input` and `joydev` (one depends on the other, remember) with a simple command, as follows:

```
matthew@seymour:~$ sudo modprobe joydev
```

If Ubuntu balks at loading a module (because it was compiled using a different kernel version from what you are currently running; for example, the NVIDIA graphics card module), you could force it to load like this:

```
matthew@seymour:~$ sudo insmod -f nvidia
```

You ignore the complaints (error messages) in this case if the kernel generates any.

When to Recompile

Ubuntu systems use a modified version of the plain-vanilla Linux kernel (a modified version is referred to as a *patched kernel*) with additional drivers and other special features compiled into it.

Ubuntu has quite an intensive testing period for all distribution kernels and regularly distributes updated versions. The supplied Ubuntu kernel is compiled with as many modules as possible to provide as much flexibility as possible. A running kernel can be further tuned with the `sysctl` program, which enables direct access to a running kernel and permits some kernel parameters to be changed. As a result of this extensive testing, configurability, and modularity, the precompiled Ubuntu kernel does everything most users need it to do. Most users only need to recompile the kernel to do the following:

- ▶ Accommodate an esoteric piece of new hardware
- ▶ Conduct a system update when Ubuntu has not yet provided precompiled kernels
- ▶ Experiment with the system capabilities

Ubuntu supplies precompiled versions of the kernel for 32- and 64-bit processors. For each architecture, Ubuntu compiles a generic kernel that works well for most uses, a server kernel that is optimized for server use, a preempt kernel designed for use in low-latency

servers, and an `rt` kernel for times when instant response is more important than balanced use (such as in professional audiovisual recording and editing). There is also a special kernel, called *virtual*, available for use in virtual machines. These are all available from the Ubuntu software repositories.

Also available are a series of packages called `linux-backports-modules-`, each with a specific set of kernel modules backported from newer mainline kernels into current version Ubuntu kernels. If you need an updated driver for a piece of hardware, look at the backported modules first.

Kernel Versions

The Linux kernel is in a constant state of development. As new features are added, bugs are fixed, and new technology is incorporated into the code base, it becomes necessary to provide stable releases of the kernel for use in a production environment. Having separate releases that contain the newest code for developers to test is also important. To keep track of the kernels, version numbers are assigned to them. Programmers enjoy using sequential version numbers that have abstract meaning. Is version 8 twice as advanced as version 4 of the same application? Is version 1 of one application less developed than version 3 of another? The version numbers cannot be used for this kind of qualitative or quantitative comparison. It is entirely possible for higher version numbers to have fewer features and more bugs than older versions. The numbers exist solely to differentiate and organize sequential revisions of software. The kernel version can be broken down into four sections:

- ▶ **Major version**—This is the major version number, which is 5 in our examples.
- ▶ **Minor version**—This is the minor version number, which is 4 in our examples.
- ▶ **Sublevel number**—This number indicates the current iteration of the kernel; which is 0 in our examples.
- ▶ **Extraversion level**—This is the number representing a collection of patches and additions made to the kernel by the Ubuntu engineers to make the kernel work for them (and you). Each collection is numbered, and the number is indicated here in the kernel name. There is not one in our preceding example. In the following example, it is 9.

Type `uname -r` at the command prompt to display your current kernel version, shown here with sample output:

```
matthew@seymour:~$ uname -r
5.4.0.9-generic
```

Obtaining the Kernel Sources

The Linux kernel has always been freely available to anyone who wants it. If you just want to recompile the existing kernel, install the `linux-source` package from the Ubuntu repositories. To get the very latest vanilla version (which is the commonly used term for the

kernel version direct from the main kernel developers and which has not yet been patched or changed by any distribution-specific kernel team), open an HTTP connection to `www.kernel.org/pub/` or use Git to retrieve from `https://git.kernel.org` or use RSYNC to retrieve from `rsync://rsync.kernel.org/pub/`. We will use kernel 5.4.0 in the following examples.

A number of different entries exist on the archive site for each kernel version, but because you are interested only in the full kernel, getting the full package of source code is necessary (for example, `linux-5.4.0.bz2`).

The `.bz2` extension is applied by the `bzip2` utility, which has better compression than `gzip`.

After it is downloaded, move the package to a directory other than `/usr/src` and unpack it. The `bzip2` unpack command is `tar -xjvf linux-5.4.0.tar.bz2`. After it is unpacked, the package creates a new directory, `linux-5.4.0`. Copy it to `/usr/src` or move it there. Then create a symbolic link of `linux-4.15` to `linux-5.4.0`. (Otherwise, some scripts will not work.) Here is how to create the symbolic link:

```
matthew@seymour:~$ sudo rm /usr/src/linux-5.4
matthew@seymour:~$ sudo ln -s /usr/src/linux-5.4.0 /usr/src/linux-5.4
```

By creating a symbolic link to `/usr/src/linux-5.4`, it is possible to allow multiple kernel versions to be compiled and tailored for different functions: You just change the symbolic link to the kernel directory you want to work on.

CAUTION

The correct symbolic link is critical to the operation of `make`. Always have the symbolic link point to the version of the kernel sources you are working with.

Patching the Kernel

Patching a kernel to the newest Linux kernel version as opposed to downloading the entire source code is possible. This choice can be beneficial for those who are not using a high-speed broadband connection. Whether you are patching existing sources or downloading the full source, the end results are identical.

Patching the kernel is not a mindless task. It requires the user to retrieve all patches from the current version to the version the user wants to upgrade to. For example, if you are currently running 5.3.3 (and have those sources) and want to upgrade to 5.4.0, you must retrieve the 5.3.3 all the way up to the 5.4.0 patch set, and so on. After you download these patches, you must apply them in succession to upgrade to 5.4.0. This is more tedious than downloading the entire source, but it's useful for those who keep up with kernel hacking and want to perform incremental upgrades to keep their Linux kernel as up to date as possible.

To patch up to several versions in a single operation, you can use the `patch-kernel` script located in the kernel source directory for the kernel version you currently use. This script applies all necessary version patches to bring your kernel up to the latest version.

The format for using the `patch-kernel` script looks like this:

```
patch-kernel source_dir patch_dir stopversion
```

The source directory defaults to `/usr/src/linux` if none is given, and `patch_dir` defaults to the current working directory if one is not supplied.

For example, assume that you have a 5.3.3 kernel code tree that needs to be patched to the 5.4.0 version. The needed earlier and 5.4.0 patch files have been downloaded and placed in the `/patch` directory in the source tree. You issue the following command in the `/usr/src/linux-5.4` directory:

```
matthew@seymour:~$ sudo scripts/patch-kernel /usr/src/linux-5.4 /usr/src/
linux-5.4.0/patch
```

Each successive patch file is applied, eventually creating a 5.4.0 code tree. If any errors occur during this operation, files named `xxx#` or `xxx.rej` are created, where `xxx` is the version of the patch that failed. You have to resolve these failed patches manually by examining the errors and looking at the source code and the patch. An inexperienced person will not have any success with this because you need to understand C programming and kernel programming to know what is broken and how to fix it. Because this was a stock 4.1.1 code tree, the patches were all successfully applied without errors. If you are attempting to apply a nonstandard third-party patch, the patch might fail.

When you have successfully patched the kernel, you are ready to begin compiling this code tree as if you were starting with a fresh, stock 5.4.0 kernel tree.

USING THE `PATCH` COMMAND

If you have a special, nonstandard patch to apply—such as a third-party patch for a commercial product—you can use the `patch` command rather than the special `patch-kernel` script that is normally used for kernel source updates. Here are some quick steps and an alternative method of creating patched code and leaving the original code alone:

1. Create a directory in your home directory and name it something meaningful, like `mylinux`.

2. Copy the pristine Linux source code there with the following:

```
cp -ravd /usr/src/linux-4.15/* ~/mylinux
```

3. Copy the patch file to that same directory as follows:

```
cp patch_filename ~/mylinux
```

4. Change to the `~/mylinux` directory with this command:

```
cd ~/mylinux
```

5. Apply the patch like this:

```
-patch -p1 < patch_filename > mypatch.log 2>&1
```

(This last bit of code saves the message output to a file so that you can look at it later.)

6. If the patch applies successfully, you are done and have not endangered any of the pristine source code. If the newly patched code does not work, you do not have to reinstall the original, pristine source code.

7. Copy your new code to `/usr/src` and make that special symbolic link described elsewhere in the chapter.

Compiling the Kernel

If you want to update the kernel from new source code you have downloaded, or if you have applied a patch to add new functionality or hardware support, you must compile and install a new kernel to actually use that new functionality. Compiling the kernel involves translating the kernel's contents from human-readable code to binary form. Installing the kernel involves putting all the compiled files where they belong in `/boot` and `/lib` and making changes to the boot loader.

The process of compiling the kernel, like the process of installing it, is almost completely automated by the `make` utility. By providing the necessary arguments and following the steps covered next, you can recompile and install a custom kernel for your use:

1. Do not delete your current kernel, so that you will have a backup that you can use to boot if there is a problem with the one you compile.
2. Apply all patches, if any, so that you have the features you desire. See the previous section for details.
3. Back up the `.config` file, if it exists, so that you can recover from the inevitable mistake. Use the following `cp` command to do so:

```
matthew@seymour:~$ sudo cp .config .config.bak
```

NOTE

If you are recompiling the Ubuntu default kernel, the `/usr/src/linux-4.15/configs` directory contains several versions of configuration files for different purposes.

Ubuntu provides a full set of `.config` files in the subdirectory `configs`, all named for the type of system they were compiled for. If you want to use one of these default configurations as the basis for a custom kernel, just copy the appropriate file to `/usr/src/linux-4.15` and rename it `.config`.

4. Run the `make mrproper` directive to prepare the kernel source tree, cleaning out any old files or binaries.
5. Restore the `.config` file that the command `make mrproper` deleted and edit the makefile to change the `EXTRAVERSION` number.

NOTE

If you want to keep any current version of the kernel that was compiled with the same code tree, manually edit the makefile with your favorite text editor and add some unique string to the `EXTRAVERSION` variable.

You can use any description you prefer.

6. Modify the kernel configuration file using `make config`, `make menuconfig`, or `make xconfig`; we recommend the last one.
7. Run `make dep` to create the code dependencies used later in the compilation process.

TIP

If you have a multiprocessor machine, you can use both processors to speed the `make` process by inserting `-jx` after the `make` command, where, as a rule of thumb, `x` is one more than the number of processors you have. You might try a larger number and even try this on a single processor machine (we have used `-j8` successfully on an SMP machine); it will only load up your CPU. Here is an example:

```
matthew@seymour:~$ sudo make -j3 bzImage
```

All the `make` processes except `make dep` work well with this method of parallel compiling.

8. Run `make clean` to prepare the sources for the compilation of the kernel.
9. Run `make bzImage` to create a binary image of the kernel.

NOTE

Several choices of directives exist; the most common ones are the following:

- ▶ **zImage**—This directive compiles the kernel, creating an uncompressed file called `zImage`.
- ▶ **bzImage**—This directive creates a compressed kernel image necessary for some systems that require the kernel image to be under a certain size for the BIOS to be able to parse them; otherwise, the new kernel will not boot. It is the most commonly used choice.

10. Run `make modules` to compile any modules your new kernel needs.
11. Run `make modules_install` to install the modules in `/lib/modules` and create dependency files.
12. Run `make install` to automatically copy the kernel to `/boot`, create any other files it needs, and modify the boot loader to boot the new kernel by default.
13. Using your favorite text editor, verify the changes made to `/etc/lilo.conf` or `/boot/grub/grub.conf`; fix if necessary and rerun `/sbin/lilo` if needed.
14. Reboot and test the new kernel.
15. Repeat the process if necessary, choosing a configuration interface.

Over time, the process for configuring the Linux kernel has changed. Originally, you configured the kernel by responding to a series of prompts for each configuration parameter; this is the `make config` utility described shortly. Although you can still configure Linux this way, most users find that type of configuration confusing and inconvenient; moving back through the prompts to correct errors, for instance, is impossible.

The `make config` utility is a command-line tool. The utility presents a question about kernel configuration options. The user responds with a `Y`, `N`, `M`, or `?`. (It is not case sensitive.) Responding `M` configures the option to be compiled as a module. A response of `?` displays

context help for that specific options, if available. (If you choose `?` and no help is available, you can turn to the vast Internet resources to find information.) We recommend that you avoid the `make config` utility.

If you prefer to use a command-line interface, you can use `make menuconfig` to configure the Linux kernel. `menuconfig` provides a graphical wrapper around a text interface. Although it is not as raw as `make config`, `menuconfig` is not a fancy graphical interface either; you cannot use a mouse but must navigate through it using keyboard commands. The same information presented in `make config` is presented by `make menuconfig`, but it looks a little nicer. Now, at least you can move back and forth in the selection process if you change your mind or make a mistake.

In `make menuconfig`, you use the arrow keys to move the selector up and down and the spacebar to toggle a selection. The Tab key moves the focus at the bottom of the screen to either Select, Exit, or Help.

If a graphical desktop is not available, `menuconfig` is the best you can do. However, both `menuconfig` and `xconfig` (see the following explanation of each) are improvements over editing the `.config` file directly. If you want to configure the kernel through a true graphical interface—with mouse support and clickable buttons—`make xconfig` is the best configuration utility option. To use this utility, you must have the X Window System running. The application `xconfig` is really nothing but a Tcl/Tk graphics widget set that provides borders, menus, dialog boxes, and the like. Its interface is used to wrap around data files that are parsed at execution time.

After loading this utility, you use it by clicking the buttons that list the configuration options. Each button you click opens another window that has the detailed configuration options for that subsection. Three buttons are at the bottom of each window: Main Menu, Next, and Prev(ious). Clicking the Main Menu button closes the current window and displays the main window. Clicking Next takes you to the next configuration section. When configuring a kernel from scratch, click the button labeled Code Maturity Level Options and then continue to click the Next button in each subsection window to proceed through all the kernel configuration choices. When you have selected all options, the main menu is again displayed. The buttons on the lower right of the main menu are for saving and loading configurations. Their functions are self-explanatory. If you just want to have a look, go exploring! Nothing will be changed if you elect not to save it.

If you are upgrading kernels from a previous release, going through the entire configuration from scratch is not necessary. Instead, you can use the directive `make oldconfig`; it uses the same text interface that `make config` uses, and it is noninteractive. It just prompts for changes for any new code.

Using `xconfig` to Configure the Kernel

For simplicity's sake, during this brisk walkthrough, this discussion assumes that you are using `make xconfig` and that prior to this point, you have completed the first five steps in the kernel compilation list shown previously.

As you learned in the preceding section, you configure the kernel using `make xconfig` by making choices in several configuration subsection windows. Each subsection window contains specific kernel options. With hundreds of choices, configuring the kernel is daunting. We cannot really offer you detailed descriptions of which options to choose because our configuration will not match your own system and setup.

Table 22.1 provides brief descriptions of many of the subsections and options so that you can get an idea of what you might encounter. We recommend that you copy your kernel's `.config` file to `/usr/src/linux-4.15` and run `make xconfig` from there. Explore all the options. As long as you do not save the file, absolutely nothing is changed on your system.

Table 22.1 Some Kernel Subsections for Configuration

Subsection	Description
Code maturity level options	Enables development code to be compiled into the kernel even if it has been marked as obsolete or as testing code only. This option should be used only by kernel developers or testers because of the possible unusable state of the code during development.
General setup	This section contains several different options covering how the kernel talks to the BIOS, whether it should support PCI or PCMCIA, whether it should use APM or ACPI, and what kind of Linux binary formats will be supported. Contains several options for supporting kernel structures necessary to run binaries compiled for other systems directly, without recompiling the program.
Loadable module support	Determines whether the kernel enables drivers and other nonessential code to be compiled as loadable modules that can be loaded and unloaded at runtime. This option keeps the basic kernel small so that it can run and respond more quickly; in that regard, choosing this option is generally a good idea.
Processor type and features	Several options dealing with the architecture that will be running the kernel.
Power management options	Options dealing with ACPI and APM power management features.
Bus options	Configuration options for the PCMCIA bus found in laptops and PCI hotplug devices.
Memory technology devices	Options for supporting flash memory devices, such as (MTD) EEPROMS. Generally, these devices are used in embedded systems.
Parallel port support	Several options for configuring how the kernel will support parallel port communications.

Subsection	Description
Plug-and-play configuration	Options for supporting <i>plug-and-play</i> (PnP) PCI, ISA, and PnP BIOS support. Generally, supporting PnP for PCI and ISA devices is a good idea.
Block devices	Section dealing with devices that communicate with the kernel in blocks of characters instead of streams. This includes IDE and ATAPI devices connected via parallel ports, as well as enabling network devices to communicate as block devices.
ATA/IDE/MFM/RLL support	Large collection of options to configure the kernel to communicate using different types of data communication protocols to talk to mass storage devices, such as hard drives. Note that this section does not cover SCSI.
SCSI device support	Options for configuring the kernel to support Small Computer System Interface (SCSI). This subsection covers drivers for specific cards, chipsets, and tunable parameters for the SCSI protocol.
Old CD-ROM drivers	Configuration options to support obscure, older CD-ROM devices that do not conform to the SCSI or IDE standards. These are typically older CD-ROM drivers that are usually a proprietary type of SCSI (not SCSI and not IDE).
Multidevice support	Options for enabling the kernel to support RAID devices in RAID and LVM software emulation and the different levels of RAID. Also contains options for support of a logical volume manager.
Fusion MPT device support	Configures support for LSI's Logic Fusion Message Passing Technology for high-performance SCSI and LAN interfaces.
IEEE1394 (firewire) support I20 device support	Experimental support for FireWire devices. Options for supporting the Intelligent Input/Output architecture, which enables the hardware driver to be split from the operating system driver, thus enabling a multitude of hardware devices to be compatible with an operating system in one implementation.
Networking support	Several options for the configuration of networking in the kernel. The options are for the types of supported protocols and configurable options of those protocols.
Amateur radio support	Options for configuring support of devices that support the AX25 protocol.
IrDA (infrared) support	Options for configuring support of the infrared Data Association suite of protocols and devices that use these protocols.
Bluetooth support	Support for the Bluetooth wireless protocol. Includes options to support the Bluetooth protocols and hardware devices.

Subsection	Description
ISDN subsystem	Options to support Integrated Services Digital Network (ISDN) protocols and devices. ISDN is a method of connection to a large area network digitally over conditioned telephone lines, largely found to connect users to ISPs.
Telephony support	Support for devices that enable the use of regular telephone lines to support Voice over Internet Protocol (VoIP) applications. This section does not handle the configuration of modems.
Input device support	Options for configuring Universal Serial Bus (USB) human interface devices (HID) such as keyboards, mice, and joysticks.
Character devices	Configuration options for devices that communicate to the server in sequential characters. This is a large subsection containing the drivers for several motherboard chipsets.
Multimedia devices	Drivers for hardware implementations of video and sound devices such as video capture boards, TV cards, and AM/FM radio adapter cards.
Graphics support	Configuration options for VGA text console, video mode selection, and support for frame buffer cards.
Sound	Large subsection to configure supported sound card drivers and chipset support for the kernel.
USB support	USB configuration options. Includes configuration for USB devices and vendor-specific versions of USB.
File system	Configuration options for supported file system types.
Additional device driver support	A section for third-party patches.
Profiling support	Profiling kernel behavior to aid in debugging and development.
Kernel hacking	A section that determines whether the kernel will contain advanced debugging options. Most users will not want to include this option in their production kernels because it increases the kernel size and slows performance by adding extra routines.
Security options	Configuration options for whether the NSA's <i>Security Enhanced Linux (SELinux)</i> is enabled.
Cryptographic options	Support for cryptography hardware. (Ubuntu patches not found in the vanilla kernel sources.)
Library routines	Options for <code>zlib</code> compression support.

After you select all the options you want, you can save the configuration file and continue with step 7 in the kernel compilation steps list shown earlier.

Creating an Initial RAM Disk Image

If you require special device drivers to be loaded to mount the root file system (for SCSI drives, network cards, or exotic file systems, for example), you must create an initial RAM disk image named `/boot/initrd.img`. For most users, creating this file is not necessary, but if you are not certain, having it really does not hurt. To create an `initrd.img` file, use the shell script `/sbin/mkinitrd`. The format for the command is the following, where `file_name` is the name of the image file you want to create:

```
/sbin/mkinitrd file_name kernel_version
```

`mkinitrd` looks at `/etc/fstab`, `/etc/modprobe.conf`, and `/etc/raidtab` to obtain the information it needs to determine which modules should be loaded during boot. For our system, we use the following:

```
matthew@seymour:~$ sudo mkinitrd initrd-5.4.0.img 5.4.0-1
```

When Something Goes Wrong

Several things might go wrong during a kernel compile and install, and several clues point to the true problem. You may see error messages printed to the screen, and some error messages are printed to the file `/var/log/syslog`, which you can examine with a text editor. If you have followed the directions for patching the kernel, you need to examine a special error log as well. Do not worry about most errors because many problems are easily fixed with some research on your part. Some errors may be unfixable, however, depending on your skill level and the availability of technical information.

Errors During Compile

Although it is rare that the kernel will not compile, there is always a chance that something has slipped through the regression testing. Let's take a look at an example of a problem that might crop up during the compile.

It is possible that the kernel compile may crash and not complete successfully, especially if you attempt to use experimental patches, add untested features, or build newer and perhaps unstable modules on an older system.

At this juncture, you have two options:

- ▶ Fix the errors and recompile.
- ▶ Remove the offending module or option and wait for the errors to be fixed by the kernel team.

Most users will be unable to fix some errors because of the complexity of the kernel code, although you should not rule out this option. It is possible that someone else discovered the same error during testing of the kernel and developed a patch for the problem, so you can check the Linux kernel mailing list archive. If the problem is not mentioned there, a search on Google might turn up something.

The second option, removing the code, is the easiest and is what most people do in cases in which the offending code is not required. In the case of the NTFS module failing, it is almost expected because NTFS support is still considered experimental and subject to errors. This is primarily because the code for the file system is reverse-engineered instead of implemented via documented standards. Read-only support has gotten better in recent kernels; write support is still experimental.

Finally, if you want to take on the task of trying to fix the problem yourself, this is a great opportunity to get involved with the Linux kernel and make a contribution that could help many others.

If you are knowledgeable about coding and kernel matters, you might want to look in the `Maintainers` file in the `/usr/src/linux-5.4/` directory of the kernel source and find the maintainer of the code. The recommended course of action is to contact the maintainer to see if the maintainer is aware of the problems you are having. If nothing has been documented for the specific error, submitting the error to the kernel mailing list is an option. The guidelines for doing this are in the `README` file in the base directory of the kernel source under the section `IF SOMETHING GOES WRONG`.

Runtime Errors, Boot Loader Problems, and Kernel Oops

Runtime errors occur as the kernel is loading. Error messages are displayed on the screen or are written to the `/var/log/syslog` file. Boot loader problems display messages to the screen; no log file is produced. Kernel oops are errors in a running kernel, and error messages are written to the `/var/log/syslog` file.

Excellent documentation exists on the Internet for troubleshooting just about every type of error that GRUB2 or the kernel could give during boot. The best way to find this documentation is to go to your favorite search engine and type in the keywords of the error you received. Adjust the keywords you use as you focus your search.

If you have GRUB problems, see the GRUB manual online at www.gnu.org/software/grub/manual/ and look for further information at <https://help.ubuntu.com/community/Grub2>.

References

- ▶ **www.kernel.org**—The Linux Kernel Archives, the source of all development discussion for the Linux kernel
- ▶ **www.gnu.org**—A source for manuals and software for programs used throughout the kernel compilation process as well as official documentation for tools such as `make` and `gcc`
- ▶ **<https://wiki.ubuntu.com/Kernel>**—The starting point for anything you want to know about both Ubuntu and its use of the Linux kernel
- ▶ **<https://wiki.ubuntu.com/Kernel/FAQ>**—The Ubuntu kernel team’s answers to the most commonly asked questions about Ubuntu’s use of Linux kernels

- ▶ **www.tldp.org**—The Linux Documentation Project, the mecca of all Linux documentation and an excellent source of HOWTO documentation as well as FAQs and online books about Linux
- ▶ **www.minix3.org**—The MINIX website, which contains a selection of links to information about MINIX, a free, open-source, operating system designed to be highly reliable, flexible, and secure

CHAPTER 23

Sharing Files and Printers

In the early days of computing, file and printer sharing was pretty much impossible because of the lack of good networking standards and interoperability. If you wanted to use a printer connected to another computer, you had to save the file to a floppy disk and walk to the other computer. Sometimes people do practically the same thing today, using a USB thumb drive or emailing the file. However, there are better ways.

Both file and printer sharing are important because it is not unusual for someone to own more than one computer. Whether you want to share photographs among various computers or have a central repository available for collaboration, file sharing is an important part of the information age. Alongside this is the need to be able to share printers; after all, people do not want to have to plug and unplug a computer to a printer just so they can print out a quick letter.

Whatever your reasons for needing to share files and printers across a network, you find out how to do both in this chapter. This chapter shows you how you can share files using the popular UNIX NFS protocol and the more Windows-friendly Samba system. The chapter covers both graphical and command-line tools, so you should find something to suit the way you work.

CAUTION

By default, Ubuntu ships with all its network ports blocked. That is, it does not listen to any requests on any network ports when it is first installed. To configure the firewall, use *Uncomplicated Firewall (UFW)* as described in Chapter 20, “Securing Your Machines.”

IN THIS CHAPTER

- ▶ Using Network File System
- ▶ Putting Samba to Work
- ▶ Network and Remote Printing with Ubuntu
- ▶ References

Using Network File System

Network File System (NFS) is a protocol developed by Sun Microsystems that enables computers to use a remote file system as if it were a real part of the local machine. A common use of NFS is to allow users' `/home` directories to appear on every local machine they use, thus eliminating the need to have physical home directories. This opens up hot desking and other flexible working arrangements, especially because no matter where the users are, their `/home` directories follow them around.

Another popular use for NFS is to share binary files between similar computers. If you have a new version of a package that you want all machines to have, you have to upgrade only on the NFS server, and all hosts running the same version of Ubuntu have the same upgraded package.

Installing and Starting or Stopping NFS

NFS is not installed by default on Ubuntu, so you need to install the `nfs-kernel-server` package. NFS itself consists of several programs that work together. One is `portmap`, which maps NFS requests to the correct daemon. Two others are `nfsd`, which is the NFS daemon, and `mountd`, which controls the mounting and unmounting of file systems.

Ubuntu automatically adds NFS to the system startup scripts, so it will always be available after you have configured it. To check this, use the command `sudo /etc/init.d/nfs-kernel-server status`, and you see that the service is running. If you need to manually start the NFS server, use the following command:

```
matthew@seymour:~$ sudo systemctl start nfs
```

In this example, NFS has been started. Use `stop` to stop the service or `restart` to restart the service. This approach to controlling NFS is handy, especially after configuration changes have been made. See the next section on how to configure NFS support on your Ubuntu system.

NFS Server Configuration

You can configure the NFS server by editing the `/etc/exports` file. This file is similar to the `/etc/fstab` file in that it is used to set the permissions for the file systems being exported. The entries look like this:

```
/file/system yourhost(options) *.yourdomain.com(options) 192.168.0.0/24(options)
```

This shows three common clients to which to share `/file/system`. The first, *yourhost*, shares `/file/system` to just one host. The second, *.yourdomain.com*, uses the asterisk (*) as a wildcard to enable all hosts in *yourdomain.com* to access `/file/system`. The third share enables all hosts of the Class C network 192.168.0.0 to access `/file/share`. For security, it is best not to use shares such as the last two across the Internet because all data will be readable by any network the data passes by.

Table 23.1 shows some common options.

Table 23.1 /etc/fstab Options

Option	Purpose
Rw	Gives read and write access
Ro	Gives read-only access
async	Writes data when the server, not the client, feels the need
Sync	Writes data as it is received

The following is an example of an `/etc/exports` file:

```
# /etc/exports: the access control list for filesystems which may be exported
#           to NFS clients.  See exports(5).
/home/matthew 192.168.0.0/24(rw,no_root_squash)
```

This file exports (makes available) `/home/matthew` to any host in `192.168.0.*` and allows users to read from and write to `/home/matthew`.

After you have finished with the `/etc/exports` file, the following command exports all the file systems in the `/etc/exports` file to a list named `xtab` under the `/var/lib/nfs` directory, which is used as a guide for mounting when a remote computer asks for a directory to be exported:

```
matthew@seymour:~$ sudo exportfs -a
```

The `-r` option, which stands for re-export, tells the command to reread the entire `/etc/exports` file and (re)mount all the entries. You can also use the `exportfs` command to export specific files temporarily. Here's an example using `exportfs` to export a file system:

```
matthew@seymour:~$ /usr/sbin/exportfs -o async yourhost:/usr/tmp
```

This command exports `/usr/tmp` to `yourhost` with the `async` option.

Be sure to restart the NFS server after making any changes to `/etc/exports`. If you prefer, you can use Ubuntu's `shares-admin` graphical client to set up NFS from the GUI. Search for "personal file sharing" in the Dash to start. Fill in the required information, and off you go. You still need to install some packages on Ubuntu for this to work (the same ones mentioned earlier in this chapter, in the "Installing and Starting or Stopping NFS" section and in the next section, "NFS Client Configuration").

NFS Client Configuration

To configure your host as an NFS client (to acquire remote files or directories), you need to ensure that you have the `nfs-common` package installed to be able to access NFS shares. After you've installed this, edit the `/etc/fstab` file as you would to mount any local file system. However, instead of using a device name to be mounted (such as `/dev/sda1`), enter the remote hostname and the desired file system to be imported. For example, an entry might look like this:

```
# Device          Mount Point  Type  Options          Freq Pass
yourhost:/home/share /export/share none  nfs 0 0
```

You can also use the `mount` command, as root, to quickly attach a remote directory to a local file system by using a remote host's name and exported directory, as in this example:

```
matthew@seymour:~$ sudo mount -t nfs 192.168.2.67:/music /music
```

After you press Enter, the entire remote directory appears on your file system. You can verify the imported file system by using the `df` command, as follows:

```
matthew@seymour:~$ df
Filesystem      1k-blocks    Used Available Use% Mounted on
/dev/hda2       18714368   9642600   8121124   55% /
/dev/hda1       46636     13247     30981    30% /boot
none           120016      0     120016    0% /dev/shm
192.168.2.67:/music 36875376 20895920 14106280  60% /music
```

Make sure the desired mount point exists before using the `mount` command. When you are finished using the directory (perhaps for copying backups), you can use the `umount` command to remove the remote file system. Note that if you specify the root directory (`/`) as a mount point, you cannot unmount the NFS directory until you reboot (because Linux complains that the file system is in use).

Putting Samba to Work

Samba uses the *Server Message Block (SMB)* protocol to enable the Windows operating system (or any other operating system) to access Linux files. Using Samba, you can make your Ubuntu machine look just like a Windows computer to other Windows computers on your network—without needing to install Windows on your PC.

Although Samba is a complex program, setting it up and using it do not have to be difficult. There are many options, which accounts for some of Samba's complexity. Depending on what you want, Samba's use can be as easy or as difficult as you would like it to be.

Fortunately, Ubuntu includes a very easy way to access files on a Windows network share by default. To start, find the Network entry in the menu at the left side of the Settings application. If there are other computers on the network with shared files using Windows or Samba, you will see a Windows Network icon that, when double-clicked, shows you all Windows domains or workgroups on your network. Just double-click any computer icon there to access shares and files.

To share from your Ubuntu desktop, right-click a folder you own while using the file browser (most likely one in your `/home/username` directory) and select Local Network Share. You are given a chance to confirm your desire to share the folder and give it a name. Sharing gives others the permission to view—but not necessarily to create or delete—files, although those permissions are also available from this menu. If you do not have the Windows sharing service already installed on your computer, Ubuntu prompts you for permission to install it.

NOTE

Most Ubuntu users do not need the information contained in the rest of this section because installing the sharing service also takes care of the configuration and, as a result, everything should just work.

For greater configurability and control, follow these instructions:

1. Install Samba and the GUI configuration application by installing these packages: `samba` and `samba-common`.
2. From the applications menu, search for `samba` and open it.
3. In the Samba configuration application, open Preferences and then Server Settings. Here you will configure basic settings such as setting Workgroup to the name of your Windows workgroup name on your network and giving a description of your computer to be seen by others on the network. You can change the computer's network security options for Samba here as well, but the default settings are good.
4. Also in the Samba configuration application, open Preferences and then Samba Users. Click Add User. You need to enter the details for the following:
 - ▶ The user on this Ubuntu machine who will be given privileges to use Samba to view files on other network computers
 - ▶ A Windows username that will be used when accessing this Ubuntu machine's files from a Windows machine on the network
 - ▶ A password that will be required with the Windows username when accessing files on this Ubuntu computer from a Windows machine
5. Restart Samba from the terminal to finish by entering `sudo service samba restart`.

You can share a folder from the Samba configuration application; set directory, name, and read/write permissions; and access users.

For even greater configurability, read through the editable configuration file at `/etc/samba/smb.conf`. It is well commented and clear and explained in greater detail later in this chapter.

To learn more about Samba, see www.samba.org. This section delves into the basics of configuring Samba, and you should first read how to manually configure Samba to get an understanding of how the software works.

When you install Samba, also installing the `samba-doc` and `samba-doc-pdf` packages is a good idea because they contain extensive documentation in text, PDF, and HTML formats. After you install Samba, you can find this documentation in `/usr/share/doc/samba*/doc`. If you install Samba using your Ubuntu disc, you can find a large amount of documentation in the directory tree starting at `/usr/share/doc/samba-doc` or `/usr/share/doc/`

samba-doc-pdf. Altogether, almost 3MB of documentation is included with the source code, in several formats, including PDF, HTML, and text.

After installing Samba, you can either create the file `/etc/samba/smb.conf` or use the `smb.conf` file supplied with Samba, which is located by default under the `/etc/samba` directory with Ubuntu. You can find nearly a dozen sample configuration files in the `/usr/share/doc/samba*/examples` directory.

NOTE

Depending on your needs, `smb.conf` can be a simple file of fewer than 20 lines or a huge file spanning many pages of text. If your needs are complex, I suggest that you browse through the Samba website at www.samba.org.

Manually Configuring Samba with `/etc/samba/smb.conf`

The `/etc/samba/smb.conf` file is broken into sections. Each section provides a description of the resource that is to be shared (called a Samba share or just a share) and should be titled appropriately. The three special sections are as follows:

- ▶ **[global]**—Establishes the global configuration settings (defined in detail in the `smb.conf` man page and Samba documentation, found in the `/usr/share/doc/samba/docs` directory)
- ▶ **[homes]**—Shares users' `/home` directories and specifies directory paths and permissions
- ▶ **[printers]**—Handles printing by defining shared printers and printer access

Each section in your `/etc/samba/smb.conf` configuration file should be named for the resource being shared. For example, if the resource `/usr/local/programs` is being shared, you could call the section `[programs]`. When Windows sees the share, it is called by whatever you name the section (`programs` in this example). The easiest and fastest way to set up this share is with the following example from `smb.conf`:

```
[programs]
path = /usr/local/programs
writeable = true
```

This bit shares the `/usr/local/programs` directory with any valid user who asks for it and makes that directory writable. It is the most basic share because it sets no limits on the directory.

Here are some parameters you can set in the sections:

- ▶ Requiring a user to enter a password before accessing a shared directory
- ▶ Limiting the hosts allowed to access the shared directory
- ▶ Altering permissions users are allowed to have on the directory
- ▶ Limiting the time of day during which the directory is accessible

The possibilities are almost endless. Any parameters set in the individual sections override the parameters set in the `[global]` section. The following section adds a few restrictions to the `[programs]` section:

```
[programs]
path = /usr/local/programs
writeable = true
valid users = mhelmke
browseable = yes
create mode = 0700
```

NOTE

You can spell it as `writeable` or `writable`; either variant will work, and both spellings are used in this chapter.

The `valid users` entry in this example limits `userid` to just `mhelmke`. All other users can browse the directory because of the `browseable = yes` entry, but only `mhelmke` can write to the directory. Any files created by `ahudson` in the directory give `ahudson` full permissions, but no one else will have access to these files. This is the same as setting permissions with the `chmod` command. Again, there are numerous options, so you can be as creative as you want when developing sections.

Setting Global Samba Behavior with the `[global]` Section

The `[global]` section establishes configuration settings for all of Samba. If a given parameter is not specifically set in another section, Samba uses the default setting in the `[global]` section. The `[global]` section also sets the general security configuration for Samba. The `[global]` section is the only section that does not require the name in brackets.

Samba assumes that anything before the first bracketed section not labeled `[global]` is part of the global configuration. (Using bracketed headings in `/etc/samba/smb.conf` makes your configuration file more readable.) The following sections discuss common Samba settings to share directories and printers. You will then see how to test your Samba configuration.

Sharing Home Directories Using the `[homes]` Section

The `[homes]` section shares Ubuntu `/home` directories for the users. The `/home` directory is shared automatically when a user's Windows computer connects to the Linux server holding the `/home` directory. The one problem with using the default configuration is that the users see all the configuration files (such as `.profile` and other files with a leading period in the filename) that they normally wouldn't see when logging on through Linux. One quick way to avoid this is to include a `path` option in the `[homes]` section. To use this solution, any user who requires a Samba share of his or her `/home` directory needs a separate "home directory" to act as the Windows `/home` directory.

This setting specifies that the directory named `share` under each user's directory is the shared Samba directory. The corresponding manual `smb.conf` setting to provide a separate "home directory" looks like this:

```
[homes]
    comment = Home Directories
    path = /home/%u/share
    valid users = %S
    read only = No
    create mask = 0664
    directory mask = 0775
    browseable = No
```

If you have a default `[homes]` section, the share shows up in the user's Network Neighborhood as the user's name. When the user connects, Samba scans the existing sections in `smb.conf` for a specific instance of the user's `/home` directory. If there is not one, Samba looks up the username in `/etc/passwd`. If the correct username and password have been given, the home directory listed in `/etc/passwd` is shared out at the user's `/home` directory. Typically, the `[homes]` section looks like this:

```
[homes]
browseable = no
writable = yes
```

The `browseable = no` entry in this example prevents other users from being able to browse the `/home` directory and is a good security practice.

This example shares out the `/home` directory and makes it writable to the user. Here's how you specify a separate Windows `/home` directory for each user:

```
[homes]
browseable = no
writable = yes
path = /path/to/windows/directories
```

Sharing Printers by Editing the `[printers]` Section

The `[printers]` section works much like the `[homes]` section but defines shared printers for use on your network. If the section exists, users have access to any printer listed in your Ubuntu `/etc/printcap` file. Note that `/etc/printcap` is not used if CUPS is installed and `printing=CUPS` is listed in the `[global]` section.

As with the `[homes]` section, when a print request is received, all the sections are scanned for the printer. If no share is found (with careful naming, there should not be a share found unless you create a section for a specific printer), the `/etc/printcap` file is scanned for the printer name that is then used to send the print request.

For printing to work properly, you must correctly set up printing services on your Ubuntu computer. A typical `[printers]` section looks like this:

```
[printers]
comment = Ubuntu Printers
```

```
browseable = no
printable = yes
path = /var/spool/samba
```

`/var/spool/samba` is a spool path set just for Samba printing.

Testing Samba with the `testparm` Command

After you have created your `/etc/samba/smb.conf` file, you can check it for correctness by using the `testparm` command. This command parses through your `/etc/samba/smb.conf` file and checks for any syntax errors. If none are found, your configuration file will probably work correctly. It does not, however, guarantee that the services specified in the file will work; it is merely making sure that the file is correctly written.

As with all configuration files, if you are modifying an existing, working file, it is always prudent to copy the working file to a different location and modify that file. Then you can check the file with the `testparm` utility. The command syntax is as follows:

```
matthew@seymour:~$ sudo testparm /path/to/smb.conf.back-up
Load smb config files from smb.conf.back-up
Processing section "[homes]"
Processing section "[printers]"
Loaded services file OK.
```

This output shows that the Samba configuration file is correct, and as long as all the services are running correctly on your Ubuntu machine, Samba should be working correctly. Now copy your old `smb.conf` file to a new location, put the new one in its place, and restart Samba with the command `sudo smbd restart`. Your new or modified Samba configuration should now be in place.

Starting, Stopping, and Restarting the `smbd` Daemon

After your `smb.conf` file is correctly configured, you might want to start, stop, or restart your Samba server daemon. You can do this with the `/usr/sbin/smbdsystemctl samba start` command, which starts the Samba server with all the defaults. The most common option you will change in this command is the location of the `smb.conf` file; you change this option if you don't want to use the default location `/etc/samba/smb.conf`. The `-s` option enables you to change the `smb.conf` file Samba uses; this option is also useful for testing whether a new `smb.conf` file actually works. Another useful option is the `-l` option, which specifies the log file Samba uses to store information.

To start, stop, or restart Samba from the command line, use the following, replacing `start` with either `stop` or `restart` as appropriate:

```
matthew@seymour:~$ sudo systemctl samba start
```

Using the `smbstatus` Command

The `smbstatus` command reports on the current status of your Samba connections. The syntax is as follows:

```
/usr/bin/smbstatus [options]
```

Table 23.2 shows some of the available options.

Table 23.2 `smbstatus` Options

Option	Result
<code>-b</code>	Brief output
<code>-d</code>	Verbose output
<code>-s /path/to/config</code>	Used if the configuration file used at startup is not the standard one
<code>-u username</code>	Shows the status of a specific user's connection
<code>-p</code>	Lists current <code>smb</code> processes, which can be useful in scripts

Connecting with the `smbclient` Command

The `smbclient` command allows users on other Linux hosts to access your `smb` shares. You cannot mount a share on your host, but you can use it in a way that is similar to an FTP client. Several options can be used with the `smbclient` command. The most frequently used is `-I` followed by the IP address of the computer to which you are connecting. The `smbclient` command does not require root access to run:

```
matthew@seymour:~$ smbclient -I 10.10.10.20 -Uusername%password
```

This gives you the following prompt:

```
smb: <current directory on share>
```

From here, the commands are almost identical to the standard UNIX/Linux FTP commands. Note that you can omit a password on the `smbclient` command line. You are then prompted to enter the Samba share password.

Mounting Samba Shares

There are two ways to mount Samba shares to your Linux host. Mounting a share is the same as mounting an available media partition or remote NFS directory except that the Samba share is accessed using SMB. The first method is to use the standard Linux `mount` command:

```
matthew@seymour:~$ sudo mount -t smbfs //10.10.10.20/homes /mount/point -o
username=sandra,dmask=777,\ fmask=777
```

NOTE

You can substitute the IP address for the hostname if your name service is running or the host is in your `/etc/hosts` file.

The preceding command mounts *sandra's* `/home` directory on your host and gives all users full permissions to the mount. The permissions are equal to the permissions on the `chmod` command.

The second method produces the same results using the `smbmount` command, as follows:

```
matthew@seymour:~$ sudo smbmount //10.10.10.20/homes /mount/point -o
username=sandra,dmask-777,\ fmask=777
```

To unmount the share, use the following standard command:

```
matthew@seymour:~$ sudo umount /mount/point
```

You can also use these `mount` commands to mount true Windows client shares to your Ubuntu host. Using Samba, you can configure your server to provide any service Windows can serve, and no one but you will ever know.

Network and Remote Printing with Ubuntu

Chapter 1, “Installing Ubuntu and Post-Installation Configuration,” discusses how to set up and configure local printers and the associated print services. This section covers configuring printers for sharing and access across a network.

Offices all around the world benefit from using print servers and shared printers. It is a simple thing to do and can bring real productivity benefits, even in small settings.

Creating Network Printers

Setting up remote printing service involves configuring a print server and then creating a remote printer entry on one or more computers on your network. This section introduces a quick method of enabling printing from one Linux workstation to another Linux computer on a LAN. You also learn about SMB printing using Samba and its utilities. Finally, this section discusses how to configure network-attached printers and use them to print single or multiple documents.

Enabling Network Printing on a LAN

If a computer with an attached printer is using Ubuntu and you want to set up the system for print serving, use the `system-config-printer` client to create a new printer, which is available in the menu at System, Administration, Printing.

You need to install any printers you have to the server as discussed in Chapter 1. (Click Add and wait a moment, and it is likely that the printer will be detected automatically; it is probably easy enough that you don't have to consult the instructions in Chapter 1.)

Next, open Server, Settings and enable Publish Shared Printers Connected to This System. Click OK. Right-click any printer's icon and select Share. That's it. Most users will not need the information in the rest of this section, even to enable access to *Common UNIX Printing System* (CUPS) via the web interface.

To enable sharing manually, edit your `/etc/cups/cupsd.conf` file. Look for the section that begins with `<Location />` and modify it so that it reads as follows:

```
<Location />
Order Deny,Allow
Deny From All
Allow From 127.0.0.1
Allow From 192.168.0.*
</Location>
```

This tells CUPS to share your printers across the network `192.168.0.*`, for example. Make sure to change this to match your network settings.

Next, you need to look in the same file for the section that starts like this:

```
Listen localhost:631
```

Modify it to show this:

```
Listen 631
```

This tells CUPS to listen on port 631 for any printer requests.

Server Message Block Printing

Printing to an SMB printer requires Samba, along with its utilities such as the `smbclient` and associated `smbprint` printing filter. You can use the Samba software included with Ubuntu to print to a shared printer on a Windows network or set up a printer attached to your system as an SMB printer. This section describes how to create a local printer entry to print to a remote shared printer using SMB.

You usually set up an SMB or shared printer under Windows operating systems through configuration settings using the Control Panel's Network device. After enabling print sharing, reboot the computer. In the My Computer, Printers folder, right-click the name or icon of the printer you want to share and select Sharing from the pop-up menu. Set the Shared As item and then enter a descriptive shared name, such as `HP2100`, and a password.

You must enter a shared name and password to configure the printer when running Linux. You also need to know the printer's workgroup name, IP address, and printer name, and you must have the username and password on hand. To find this information, select Start, Settings, Printers, and then right-click the shared printer's listing in the Printers window. Select Properties from the pop-up window.

You can use CUPS to configure Samba to use your printers by editing the `smb.conf` file.

In the `[global]` section enter the following lines if they are not already there:

```
...
load printers = yes
printing = cups
printcap name = cups
```

This tells Samba to use CUPS to provide printing services. Next, you need to create a new section in the `smb.conf` file at the end of the file, as follows:

```
[printers]
comment = Use this for All Printers
path = /var/spool/samba
browseable = no
public = yes
guest ok = yes
writable = no
printable = yes
printer admin = root, andrew
```

This publishes your printers to the network and allows others to connect to them via Windows clients.

Make sure you restart the Samba service using the command shown earlier to cause Samba to pick up the changes to the configuration file.

Using the CUPS GUI

You can use CUPS to create printer queues, get print server information, and manage queues by launching a browser (such as Firefox) and browsing to `http://localhost:631`. CUPS provides a web-based administration interface, as shown in Figure 23.1.



FIGURE 23.1 Use the web-based CUPS administrative interface to configure and manage printing.

If you click the Administration tab in the browser page, you can start configuring your printers, as shown in Figure 23.2.

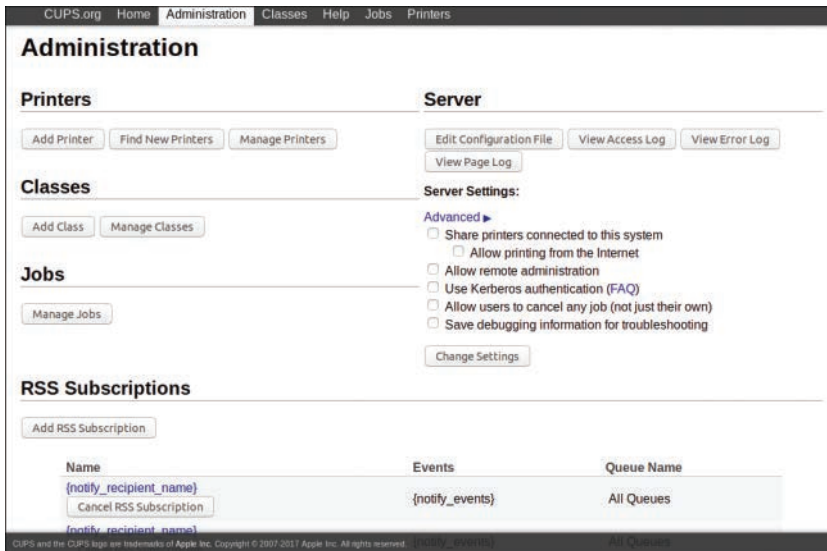


FIGURE 23.2 Select the Administration tab to perform printer administration with CUPS.

Creating a CUPS Printer Entry

This section provides a short example of creating a Linux printer entry using the CUPS web-based interface. You can use the CUPS interface to create a printer and device queue type (such as local, remote, serial port, or Internet); then you can enter a device *uniform resource identifier (URI)*, such as `lpd://192.168.2.35/lp`, which represents the IP address of a remote UNIX print server and the name of the remote print queue on the server. You also need to specify the model or make of printer and its driver. A Printers page link enables you to print a test page, stop the printing service, manage the local print queue, modify the printer entry, or add another printer.

In the Admin page, click the Add Printer button and then enter the username and password of someone on this computer who has sudo privileges. You can then select from detected local printers, discovered network printers, or the type of printer for which you may want to enter details manually.

CUPS offers many additional features and provides transparent traditional UNIX printing support for Ubuntu.

NOTE

To learn more about CUPS and to get a basic overview of the system, visit www.cups.org.

Avoiding Printer Support Problems

Troubleshooting printer problems can be frustrating, especially if you find that your new printer is not working properly with Linux. Keep in mind, however, that nearly all

printers on the market today work with Linux. However, some vendors have higher batting averages than others in the game of supporting Linux.

All-in-One (Print/Fax/Scan) Devices

Problematic printers, or printing devices that might or might not work with Ubuntu, include multifunction (or *all-in-one*) printers that combine scanning, faxing, and printing services. You should research any planned purchase and avoid any vendor that is unwilling to support Linux with drivers or development information.

Using USB and Legacy Printers

Problems can arise because of a lack of a printer's *Universal Serial Bus (USB)* vendor and device ID information—a problem shared by some USB scanners under Linux.

RELATED UBUNTU AND LINUX COMMANDS

The following commands can help you manage printing services:

- ▶ **cupsaccept**—Allows print job access to the CUPS server
- ▶ **cupsreject**—Prevents print job access to the CUPS server
- ▶ **cupscancel**—Cancels a print job
- ▶ **cupsdisable**—Disables printing
- ▶ **cupsenable**—Enables printing
- ▶ **lp**—Sends a specified file to the printer and allows control of the print service
- ▶ **lpc**—Displays the status of printers and print service at the console
- ▶ **lpq**—Views print queues (pending print jobs) at the console
- ▶ **lprm**—Removes print jobs from the print queue via the command line
- ▶ **lpstat**—Displays printer and server status

References

- ▶ **<https://help.ubuntu.com/community/SettingUpNFHowTo>**—Ubuntu community documentation for setting up NFS
- ▶ **<https://help.ubuntu.com/community/Samba>**—Ubuntu community documentation for setting up Samba
- ▶ **www.samba.org**—Base entry point for getting more information about Samba and using the SMB protocol with Linux, UNIX, macOS, and other operating systems
- ▶ **www.cups.org**—A comprehensive repository of CUPS software and information

This page intentionally left blank

CHAPTER 24

Common Web Server Stacks

IN THIS CHAPTER

- ▶ LAMP
- ▶ LEMP
- ▶ MEAN
- ▶ References

A *web server stack* is a set of software installed and configured to work together to serve content over a network. There are many stacks of software that are common enough to be referred to using acronyms. This chapter describes some of the most popular. Before diving deeply into each of the related technologies, this chapter introduces some of the most popular types of web server stacks in use today.

All the stacks in this chapter can be assembled and run on Ubuntu or any Linux distribution and are common enough to be worth knowing something about. The pieces being used to assemble the stacks described in this chapter can often be replaced with different software that serves the same purpose, as is illustrated in the discussion of the LAMP stack.

There is a lot to think about, regardless of which stack you choose. This chapter is intended to give a high-level overview of a few popular options. Large books have been written and are available for the various stacks, for each of the underlying layers in the stacks, and so on. The goal of this chapter is to help you begin to think about how to accomplish a much larger goal, not to outline all the necessary steps and configurations.

LAMP

LAMP was one of the first stacks to become popular. It is composed of Linux, the Apache web server, the MySQL database, and either PHP, Python, or Perl. (See the bonus online chapters for more information about these languages.) Because all the technologies comprising LAMP are free and open source, LAMP is inexpensive to set up and

use, making it much more affordable than other options for setting up a web server or even selling web hosting to others.

LAMP became popular during the initial rise of Linux as an operating system for web hosting, both in large enterprise corporations looking to save money and in small startup companies selling server space and web hosting to mom-and-pop stores wanting to join the Internet craze. With its careful concern for stability and security, inherited from the creator's love of UNIX, coupled with being free to install and use and update, Linux was from the start an ideal platform for most hosting needs.

LAMP is still popular and in wide use today. Many large sites use LAMP applications, and LAMP is not going away anytime soon.

Running on top of Linux is the Apache web server. (See Chapter 25, "Apache Web Server Management," for detailed information about Apache.) Apache takes care of listening to an open communication port for requests and responding to those requests by serving the requested information.

Working in conjunction with Apache is a database that stores the information to be served. This is typically a relational database such as MySQL. (See Chapter 28, "Administering Relational Database Services," for detailed information about MySQL.) These days, it could be MySQL or MariaDB or PostgreSQL or another similar product.

Off-the-shelf commodity as well as custom content management software has been written using PHP, Python, or Perl. For example, WordPress uses PHP, django CMS uses Python, and Movable Type uses Perl. Which language you install and use depends on the framework or software you are using or which you prefer to use to write your software.

Of the web stacks discussed in this chapter, the LAMP stack is the easiest to install quickly. If you install Ubuntu Server, you are given an option of setting up the machine as a LAMP server during installation. You can even set the root password for MySQL during installation. The upside is that you can have a complete LAMP server up and running in 15 minutes or so. The downside to this method is that you are limited in your configuration. You will install Ubuntu (Linux), Apache, MySQL, and PHP. That is great if you simply want to install a product such as WordPress next or do some simple web development. For anything more unique, however, we suggest that you install the server first and then install and configure each component, one at a time.

To install LAMP on your own, you need to first install Ubuntu—which can be standard Ubuntu if you are going to use a local laptop or desktop machine for development or Ubuntu Server on a remote machine. Then you can install each of the following components, setting up and configuring each one before moving to the next:

- 1.** Install OpenSSH and configure the machine for SSH access. This is vital if you intend to eventually serve content via HTTPS, which is becoming a necessity as major browsers like Google Chrome intend to mark as insecure any websites that do not use HTTPS.
- 2.** Install Apache 2 and set it up for the sites you intend to host, including HTTPS configuration. Later, you will also configure your firewall to allow traffic via the ports specific to HTTP and HTTPS (80 and 443).

As a side note, unless you are a large corporation, you do not need to pay for web certificates from an expensive Certificate Authority anymore. Check out Let's Encrypt at <https://letsencrypt.org/>.

3. Install MySQL, MariaDB, or your relational database of choice, making sure to set a secure root password. You are strongly urged to create a web server-specific user in the database that will interact with the database rather than use the root account in your software.
4. Install PHP, Python, or Perl as appropriate for the software you intend to run/write that will display your content.

Alternatively, you can install and set up a basic LAMP stack much more easily using a couple commands, as follows:

```
$ sudo apt install tasksel
$ sudo tasksel install lamp-server
```

This will install everything mentioned previously interactively, giving you a chance to set your MySQL root password and so on during the process. Using `tasksel` is much more convenient, but with a generic and simple configuration.

At this point, regardless of which method you use, the entire stack is installed. You can now proceed to install open source software such as Drupal or develop your own.

LEMP

The main difference between LAMP and LEMP is the web server that is used. Whereas LAMP uses Apache, LEMP uses Nginx. (See Chapter 26, “Nginx Web Server Management,” for detailed information about Nginx.)

NOTE

It seems that whoever created the acronym LEMP decided a vowel was needed, so the *E* stands for *Nginx*—but the server name is pronounced as though it starts with an *E*, so it actually makes sense.

Beyond the obvious differences between Apache and Nginx as mature web servers with different use cases, the only other difference is that there is currently not a LEMP option in the Ubuntu Server installation. That does not make much difference, though, as we recommend that you install Ubuntu first and then install and configure each component individually rather than using the shortcut during Ubuntu installation.

NOTE

We could probably write a similar section for nearly every HTTP server listed in Chapter 27, “Other HTTP Servers.” If someone wants to install an LLMP stack (Linux, `lighttpd`, MySQL, PHP), for example, it is certainly possible.

MEAN

As the web moved forward with more and more dynamic websites and applications, it became vital for web developers to know JavaScript. Then someone had the idea of writing applications that could run both on the client *and* server sides in the same language. The foundational reasoning for doing this was speed. It takes time to develop for the Web. If you use multiple programming languages, you have to either learn multiple languages or hire people who know the languages you use. Further, as more and more websites have become fully interactive experiences, learning JavaScript is already a necessity for more developers. Therefore, JavaScript is now the language of choice for both the client and server sides—and it is an important component in the MEAN web stack.

You do not have to be running Linux to serve content with MEAN. You can also run a MEAN stack on Windows or on macOS. In addition, everything for this stack is written in JavaScript. The database used is one of the NoSQL options, MongoDB, which works very well with the JSON data format used in a MEAN stack.

A MEAN stack consists of four main components, regardless of the operating system being used as a platform:

- ▶ **MongoDB**—One of the NoSQL databases discussed in Chapter 29, “NoSQL Databases,” MongoDB is used to store data in JSON format for an application.
- ▶ **Express.js**—This back-end web application framework runs on Node.js.
- ▶ **AngularJS or Angular**—These are both JavaScript MVC (model–view–controller) frameworks. Angular is newer and is a complete rewrite of AngularJS, with some intentional changes along the way. Angular provides the front-end web application framework that allows you to run code in a user’s browser.
- ▶ **Node.js**—Node.js is an execution environment for event-driven server-side applications.

Regardless of your operating system, most MEAN stack users recommend downloading and installing components from the respective websites rather than by using a package manager provided by your operating system. Websites for each are listed here; download and install the software in this order, following the provider’s instructions:

1. Install MongoDB from www.mongodb.com/download-center.
2. Install Node.js from <https://nodejs.org/en/>. When you do, you also install the `npm` package manager, which you will use next.
3. Use the command `npm install express` to install Express.js.
4. Follow the Getting Started guide at <https://angular.io> to install Angular.

What is really different in a MEAN application than in a LAMP or other traditional application is the flexibility. You can run application code on your server. You can run code in the user’s browser. You can even run some application logic in the database, which is especially nice for some types of analytics.

NOTE

You can replace Angular (originally from Google) with React (from Facebook), which serves a similar purpose. You can also replace it with other frameworks, such as Ember.js or Backbone.js, and still do all your development in JavaScript. Some love this flexibility, but to others it is overwhelming. In any case way, MEAN development is here and is not likely to disappear anytime soon.

References

- ▶ <https://help.ubuntu.com/lts/serverguide/lamp-overview.html>—LAMP in the Ubuntu Server Guide
- ▶ www.mongodb.com—The official site of MongoDB
- ▶ <https://nodejs.org>—The official site of Node.js
- ▶ www.npmjs.com—Home of the `npm` package manager
- ▶ <https://expressjs.com>—The official site of Express.js
- ▶ <https://angular.io>—The official site of Angular
- ▶ <https://emberjs.com>—The official site of Ember.js
- ▶ <http://backbonejs.org>—The official site of Backbone.js
- ▶ <https://reactjs.org>—React, a JavaScript library for building user interfaces

This page intentionally left blank

CHAPTER 25

Apache Web Server Management

This chapter covers the configuration and management of the Apache web server and includes an overview of some of the major components of the server and discussions of text-based and graphical server configuration. In this chapter, you learn how to start, stop, and restart Apache using the command line. The chapter begins with some introductory information and then shows you how to install, configure, and use Apache.

About the Apache Web Server

Apache is the most widely used web server on the Internet today. The name *Apache* appeared during the early development of the software because it was “a patchy” server, made up of patches for the freely available source code of the NCSA HTTPd web server. For a while after the NCSA HTTPd project was discontinued, a number of people wrote a variety of patches for the code to either fix bugs or add features they wanted. A lot of this code was floating around, and people were freely sharing it, but it was completely unmanaged.

After a while, Brian Behlendorf and Cliff Skolnick set up a centralized repository of these patches, and the Apache project was born. The project still involves a small core group of programmers, but anyone is welcome to submit patches to the group for possible inclusion in the code.

There has been a surge of interest in the Apache project over the past several years, partially buoyed by a new interest in open source on the part of enterprise-level information services. It’s also due in part to crippling security flaws in Microsoft’s *Internet Information Services (IIS)*; the existence of malicious web task exploits; and operating system and

IN THIS CHAPTER

- ▶ About the Apache Web Server
- ▶ Installing the Apache Server
- ▶ Runtime Server Configuration Settings
- ▶ File System Authentication and Access Control
- ▶ Apache Modules
- ▶ Virtual Hosting
- ▶ Logging
- ▶ HTTPS
- ▶ References

networking vulnerabilities to the now-infamous Code Red, Blaster, and Nimda worms. IBM made an early commitment to support and use Apache as the basis for its web offerings and has dedicated substantial resources to the project because using an established, proven web server makes sense.

In mid-1999, the Apache Software Foundation (ASF) was incorporated as a nonprofit company. A board of directors, who are elected on an annual basis by the ASF members, oversees the company. This company provides a foundation for several open source software development projects, including the Apache web server project.

TIP

You can find an overview of Apache in its FAQs at <https://wiki.apache.org/httpd/FAQ>. In addition to extensive online documentation, you'll find the complete documentation for Apache in the HTML directory of your Apache server. If you have Apache running on your system, you can access this documentation by looking at <https://localhost/manual/index.html>.

To determine the precise version of Apache included with your system, use the following:

```
matthew@seymour:~$ apache2 -v
```

Installing the Apache Server

Install the `apache2` package from the Ubuntu software repositories. Updated packages usually contain important bug and security fixes. When an updated version is released, install it as quickly as possible to keep your system secure.

NOTE

Check the Apache site for security reports. Browse to https://httpd.apache.org/security_report.html for links to security vulnerabilities for Apache 2.0, 2.2, and 2.4. Subscribe to a support list or browse through up-to-date archives of all Apache mailing lists at <https://httpd.apache.org/mail/> for various articles or <https://httpd.apache.org/lists.html> for comprehensive and organized archives.

CAUTION

Be wary of installing experimental packages and never install them on production servers (that is, servers used in “real life”). Very carefully test packages beforehand on a host that is not connected to a network!

For more information about installing software from the Ubuntu repositories, see Chapter 9, “Managing Software.”

Starting and Stopping Apache

At this point, you should have installed your Apache server with its default configuration. Ubuntu provides a default home page at `/var/www/html/index.html` as a test.

You can start Apache from the command line of a text-based console or X terminal window, and you must have root permission to do so. How you do so depends on the release version of Ubuntu that you are running. For Ubuntu 16.04 and later, you use `systemd` commands. For earlier Ubuntu releases like 12.04 and 14.04 that used Upstart, you use Upstart commands. Some prefer to use `apache2ctl` commands, which work across most distributions. Table 25.1 compares the commands.

Table 25.1 Comparing Commands Used in `systemd`, Upstart, and `apache2ctl`

Action	<code>systemd</code>	Upstart	<code>apache2ctl</code>
Start	<code>sudo systemctl start apache2.service</code>	<code>sudo start apache2</code>	<code>sudo apache2ctl start</code>
Stop	<code>sudo systemctl stop apache2.service</code>	<code>sudo stop apache2</code>	<code>sudo apache2ctl stop</code>
Restart	<code>sudo systemctl restart apache2.service</code>	<code>sudo restart apache2</code>	<code>sudo apache2ctl restart</code>

The server daemon, `apache2`, recognizes several command-line options you can use to set some defaults, such as specifying where `apache2` reads its configuration directives. The Apache `apache2` executable also understands other options that enable you to selectively use parts of its configuration file, specify a different location for the actual server and supporting files, use a different configuration file (perhaps for testing), and save startup errors to a specific log. The `-v` option causes Apache to print its development version and quit. The `-v` option shows all the settings that were in effect when the server was compiled.

The `-h` option prints the following usage information for the server:

```
matthew@seymour:~$ apache2 -h
Usage: apache2 [-D name] [-d directory] [-f file]
               [-C "directive"] [-c "directive"]
               [-k start|restart|graceful|stop]
               [-v] [-V] [-h] [-l] [-L] [-t]

Options:
  -D name           : define a name for use in <IfDefine name> directives
  -d directory     : specify an alternate initial ServerRoot
  -f file          : specify an alternate ServerConfigFile
  -C "directive"   : process directive before reading config files
  -c "directive"   : process directive after reading config files
  -e level         : show startup errors of level (see LogLevel)
  -E file          : log startup errors to file
  -v              : show version number
  -V              : show compile settings
  -h              : list available command line options (this page)
  -l              : list compiled in modules
  -L              : list available configuration directives
  -t -D DUMP_VHOSTS : show parsed settings (currently only vhost settings)
  -t              : run syntax check for config files
```

Other options include listing Apache's *static modules*, or special, built-in independent parts of the server, along with options that can be used with the modules. These options, called *configuration directives*, are commands that control how a static module works. Note that Apache also includes a large number of *dynamic modules*, or software portions of the server that can be optionally loaded and used while the server is running.

The `-t` option is used to check your configuration files. Running this check before restarting your server is a good idea, especially if you've made changes to your configuration files. Such tests are important because a configuration file error can result in your server shutting down when you try to restart it. There is a bug in the internal username settings for `apache2` in Ubuntu that gives you this error if you enter the following:

```
matthew@seymour:~$ sudo apache2 -t
apache2: bad user name ${APACHE_RUN_USER}
```

If this happens to you, enter the command this way to force the command to use the expected username settings, and you will get the proper output:

```
matthew@seymour:~$ sudo APACHE_RUN_USER=www-data APACHE_RUN_GROUP=www-data
apache2 -t
```

Runtime Server Configuration Settings

At this point, the Apache server will run, but perhaps you want to change a behavior, such as the default location of your website's files. This section covers the basics of configuring the server to work the way you want it to work.

Runtime configurations are stored in just one file—`apache2.conf`, which is under the `/etc/apache2` directory. You can use this configuration file to control the default behavior of Apache, such as the web server's base configuration directory (`/etc/apache2`), the name of the server's PID file (`/var/run/apache2.pid`), or its response timeout (300 seconds). Apache reads the data from the configuration file when started (or restarted).

Runtime Configuration Directives

You perform runtime configuration of your server with configuration directives, which are commands that set options for the `apache2` daemon. The directives are used to tell the server about various options you want to enable, such as the location of files important to the server configuration and operation. Apache supports nearly 300 configuration directives, using the following syntax:

```
directive option option...
```

Each directive is specified on a single line. See the following sections for some examples of directives and how to use them. A directive may only set a value such as a filename, or a directive may enable you to specify various options. Some special directives, called *section directives*, look like HTML tags. Section directives are surrounded by angle brackets, such as

`<Directory>`. Section directives usually enclose a group of directives that apply only to the directory specified in the section:

```
<Directory somedir/in/your/tree>
    directive option option
    directive option option
</Directory>
```

A section is always closed with a matching section tag that looks like this: `</Directory>`. Note that section tags, like any other directives, are specified one per line.

TIP

Apache is configured with an alias that lets you view the documentation installed in `/usr/share/doc` by using your web browser at `localhost/manual`. After installing and starting Apache, you can find an index of directives at <https://localhost/manual/mod/directives.html>.

Editing `apache2.conf`

Most of the default settings in the `config` file are okay to keep, particularly if you've installed the server in a default location and aren't doing anything unusual on your server. The file includes clear comments describing most of the settings. In general, if you do not understand what a particular directive is for, leave it set to the default value.

The following sections describe some of the configuration file settings you *might* want to change concerning operation of your server.

ServerRoot

The `ServerRoot` directive sets the absolute path to your server directory. This directive tells the server where to find all the resources and configuration files. Many of these resources are specified in the configuration files relative to the `ServerRoot` directory.

Your `ServerRoot` directive should be set to `/etc/apache2` if you installed the Ubuntu package or `/usr/local/apache` (or whatever directory you chose when you compiled Apache) if you installed from the source. This is commented out in the file, but `apache2 -v` shows that this default has been compiled into the package.

Listen

The `Listen` directive is actually in a file called `ports.conf` that is included from `apache2.conf` and indicates on which port you want your server to run. By default, this is set to 80, which is the standard HTTP port number. You might want to run your server on another port—for example, when running a test server that you don't want people to find by accident. Do not confuse this with real security! See the “File System Authentication and Access Control” section, later in this chapter, for more information about how to secure parts of your web server.

User and Group

The `User` and `Group` directives should be set to the UID and GID the server will use to process requests.

In Ubuntu, set these configurations to a user with few or no privileges. In this case, they're set to user `www-data` (a user defined specifically to run Apache) and group `www-data`. If you want to use a different UID or GID, be aware that the server will run with the permissions of the user and group set here. This means that, in the event of a security breach, whether on the server or (more likely) in your own CGI programs, those programs will run with the assigned UID. If the server runs as root or some other privileged user, someone can exploit the security holes and do nasty things to your site. Always think in terms of the specified user running a command such as `rm -rf /` because that would wipe all files from your system. This should convince you that leaving `www-user` as a user with no privileges is probably a good thing.

Instead of specifying the `User` and `Group` directives using names, you can specify them using the UID and GID numbers. If you use numbers, be sure that the numbers you specify correspond to the user and group you want and that they're preceded by the pound (#) symbol.

Here's how these directives look if specified by name:

```
User www-user
Group www-user
```

Here are the same directives specified using UID and GID:

```
User #48
Group #48
```

TIP

If you find a user on your system (other than root) with a UID and GID of 0, your system has been compromised by a malicious user.

ServerAdmin

The `ServerAdmin` directive should be set to the address of the webmaster managing the server. This address should be a valid email address or alias, such as `webmaster@matthewhelmke.com`, because this address is returned to a visitor when a problem occurs on the server.

ServerName

The `ServerName` directive sets the hostname the server will return. Set it to a *fully qualified domain name (FQDN)*. For example, set it to `www.your.domain` rather than simply `www`. This is particularly important if this machine will be accessible from the Internet rather than just on your local network.

You do not need to set the `ServerName` directive unless you want a name other than the machine's canonical name returned. If this value isn't set, the server will figure out the

name by itself and set it to its canonical name. However, you might want the server to return a friendlier address, such as `www.your.domain`. Whatever you do, `ServerName` should be a real *Domain Name System (DNS)* name for your network. If you're administering your own DNS, remember to add an alias for your host. If someone else manages the DNS for you, ask that person to set this name for you.

DocumentRoot

Set the `DocumentRoot` directive to the absolute path of your document tree, which is the top directory from which Apache will serve files. By default, it's set to `/var/www/`. If you built the source code yourself and did not choose another directory when you compiled Apache, `DocumentRoot` is set to `/usr/local/apache/htdocs`.

UserDir

The `UserDir` directive disables or enables and defines the directory (relative to a local user's `/home/username` directory) where that user can put public HTML documents. It is relative because each user has her own HTML directory. This setting is disabled by default but can be enabled to store user web content under any directory.

The default setting for this directive, if enabled, is `public_html`. Each user can create a directory called `public_html` under her `/home` directory, and HTML documents placed in that directory are available as `http://servername/~username`, where `username` is the username of the particular user.

DirectoryIndex

The `DirectoryIndex` directive indicates which file should be served as the index for a directory, such as which file should be served if the URL `http://servername/SomeDirectory/` is requested.

Putting a list of files here is often useful so that if `index.html` (the default value) isn't found, another file can be served instead. The most useful application of this is to have a CGI program run as the default action in a directory. If you have users who make their web pages on Windows, you might want to add `index.htm` as well. In that case, the directive looks like `DirectoryIndex index.html index.cgi index.htm`.

Apache Multiprocessing Modules

Apache version 2.0 and later now uses a new internal architecture that supports *multiprocessing modules (MPMs)*. These modules are used by the server for a variety of tasks, such as network and process management, and are compiled into Apache. MPMs enable Apache to work much better on a wider variety of computer platforms, and they can help improve server stability, compatibility, and scalability.

Apache can use only one MPM at any time. These modules are different from the base set included with Apache (see the "Apache Modules" section later in this chapter) but are used to implement settings, limits, or other server actions. Each module in turn supports numerous additional settings, called *directives*, which further refine server operation.

The internal MPM modules relevant for Linux include the following:

- ▶ **mpm_common**—A set of 20 directives common to all MPM modules
- ▶ **prefork**—A nonthreaded, preforking web server that works much like earlier (1.3) versions of Apache
- ▶ **worker**—A hybrid multiprocess multithreaded server

MPM enables Apache to be used on equipment with fewer resources yet still handle massive numbers of hits and provide stable service. The `worker` module provides directives to control how many simultaneous connections your server can handle.

NOTE

Other MPMs related to other platforms are available for Apache, such as `mpm_netware` for NetWare hosts and `mpm_winnt` for NT platforms. An MPM named `perchild`, which provides user ID assignment to selected daemon processes, is under development. For more information, browse to the Apache Software Foundation's home page at www.apache.org.

Using `.htaccess` Configuration Files

Apache supports special configuration files, known as `.htaccess` files. Almost any directive that appears in `apache2.conf` can appear in an `.htaccess` file. This file, specified in the `AccessFileName` directive in `apache2.conf`, sets configurations on a per-directory (usually in a user directory) basis. As the system administrator, you can specify both the name of this file and which of the server configurations can be overridden by the contents of this file. This is especially useful for sites in which there are multiple content providers, and you want to control what these people can do with their space.

To limit which server configurations the `.htaccess` files can override, use the `AllowOverride` directive. `AllowOverride` can be set globally or per directory. For example, in your `apache2.conf` file, you could use the following:

```
# Each directory to which Apache has access can be configured with respect
# to which services and features are allowed and/or disabled in that
# directory (and its subdirectories).
#
# First, we configure the "default" to be a very restrictive set of
# permissions.
#
<Directory />
    Options FollowSymLinks
    AllowOverride None
</Directory>
```

Options Directives

To configure which configuration options are available to Apache by default, you must use the `Options` directive. Options can be `None`; `All`; or any combination of `Indexes`, `Includes`, `FollowSymLinks`, `ExecCGI`, and `MultiViews`. `MultiViews` is not included in `All` and must be specified explicitly. These options are explained in Table 25.2.

Table 25.2 Switches Used by the `Options` Directive

Switch	Description
<code>None</code>	None of the available options are enabled for this directory.
<code>Indexes</code>	In the absence of an <code>index.html</code> file or another <code>DirectoryIndex</code> file, a listing of the files in the directory is generated as an HTML page for display to the user.
<code>Includes</code>	<i>Server-side includes</i> (SSIs) are permitted in this directory. This can also be written as <code>IncludesNoExec</code> if you want to allow includes but don't want to allow the <code>exec</code> option in them. For security reasons, this is usually a good idea in directories over which you don't have complete control, such as <code>UserDir</code> directories.
<code>FollowSymLinks</code>	This option allows access to directories that are symbolically linked to a document directory. You should never set this globally for the whole server and only rarely set it for individual directories. This option is a potential security risk because it allows web users to escape from the document directory and could potentially allow them access to portions of your file system where you really don't want people poking around.
<code>ExecCGI</code>	CGI programs are permitted in this directory, even if it is not a directory defined in the <code>ScriptAlias</code> directive.
<code>MultiViews</code>	This is part of the <code>mod_negotiation</code> module. When a client requests a document that can't be found, the server tries to figure out which document best suits the client's requirements.

NOTE

These directives also affect all subdirectories of the specified directory.

AllowOverrides Directives

The `AllowOverrides` directives specify which configuration options `.htaccess` files can override. You can set `AllowOverrides` individually for each directory. For example, you can have different standards about what can be overridden in the main document `root` and in `UserDir` directories. This capability is particularly useful for user directories, where the user does not have access to the main server configuration files.

`AllowOverrides` can be set to `All` or any combination of `Options`, `FileInfo`, `AuthConfig`, and `Limit`. These options are explained in Table 25.3.

Table 25.3 Switches Used by the AllowOverrides Directive

Switch	Description
Options	The <code>.htaccess</code> file can add options not listed in the <code>Options</code> directive for this directory.
FileInfo	The <code>.htaccess</code> file can include directives for modifying document type information.
AuthConfig	The <code>.htaccess</code> file might contain authorization directives.
Limit	The <code>.htaccess</code> file might contain <code>allow</code> , <code>deny</code> , and <code>order</code> directives.

File System Authentication and Access Control

You're likely to include material on your website that isn't supposed to be available to the public. You must be able to lock out this material from public access and provide designated users with the means to unlock the material. Apache provides two methods for accomplishing this type of access: authentication and authorization. You can use different criteria to control access to sections of your website, including checking the client's IP address or hostname or requiring a username and password. This section briefly covers some of these methods.

CAUTION

Allowing individual users to put web content on your server poses several important security risks. If you're operating a web server on the Internet rather than on a private network, check out www.w3.org/Security/.

Restricting Access with `Require`

One of the simplest ways to limit access to website material is to restrict access to a specific group of users, based on IP addresses or hostnames. Apache uses the `Require` directive to accomplish this. Here are some examples with comments that could be placed within the `apache2.conf` file:

```
<RequireAll>
    Require all granted                #permit all to access
    Require not ip 10.252.46.163      #except from this ip address
    Require not host horriblepeople.com #and also not from this domain
    Require not host gov              #and finally, not from any .gov
</RequireAll>
```

There are many options beyond `RequireAll`, including `RequireAny` and `RequireNone`, along with a detailed set of options for each. For more, see <https://httpd.apache.org/docs/2.4/howto/access.html>.

Authentication

Authentication is the process of ensuring that visitors really are who they claim to be. You can configure Apache to allow access to specific areas of web content only to clients who can authenticate their identity. There are several methods of authentication in Apache; Basic Authentication is the most common (and the method discussed in this chapter).

Under Basic Authentication, Apache requires a user to supply a username and a password to access the protected resources. Apache then verifies that the user is allowed to access the resource in question. If the username is acceptable, Apache verifies the password. If the password also checks out, the user is authorized, and Apache serves the request.

HTTP is a stateless protocol; each request sent to the server and each response is handled individually—and not in an intelligent fashion. Therefore, the authentication information must be included with each request. This means each request to a password-protected area is larger and therefore somewhat slower. To avoid unnecessary system use and delays, protect only those areas of your website that absolutely need protection.

To use Basic Authentication, you need a file that lists the users who are allowed to access the resources. This file is composed of a plain-text list containing name and password pairs. It looks very much like the `/etc/passwd` user file of your Linux system.

CAUTION

Do not use `/etc/passwd` as a user list for authentication. When you're using Basic Authentication, passwords and usernames are sent as Base64-encoded text (which is just as readable as plain text) from the client to the server. The username and password are included in each request that is sent to the server. So, anyone who might be snooping on your traffic would be able to get this information!

To create a user file for Apache, use the `htpasswd` command that is included with the Apache package. Running `htpasswd` without any options produces the following output:

Usage:

```
htpasswd [-cmdps] passwordfile username
htpasswd -b[cmdps] passwordfile username password
```

```
htpasswd -n[mdps] username
htpasswd -nb[mdps] username password
```

- c Create a new file.
- n Don't update file; display results on stdout.
- m Force MD5 encryption of the password.
- d Force CRYPT encryption of the password (default).
- p Do not encrypt the password (plaintext).
- s Force SHA encryption of the password.
- b Use the password from the command line rather than prompting for it.
- D Delete the specified user.

On Windows, TPF and NetWare systems the '-m' flag is used by default.

On all other systems, the '-p' flag will probably not work.

As you can see, it is not a difficult command to use. For example, to create a new user file named `gnulixusers` with a user named `wsb`, you need to do something like this:

```
matthew@seymour:~$ sudo htpasswd -c gnulixusers wsb
```

You are then prompted for a password for the user. To add more users, repeat the same procedure but without the `-c` flag.

You can also create user group files. The format of these files is similar to that of `/etc/groups`. On each line, enter the group name, followed by a colon (`:`), and then list all users, with users separated by spaces. For example, an entry in a user group file might look like this:

```
gnulixusers: wsb pgj jp ajje nadia rkr hak
```

Now that you know how to create a user file, it's time to look at how Apache might use such a file to protect web resources.

To point Apache to the user file, use the `AuthUserFile` directive. `AuthUserFile` takes the file path to the user file as its parameter. If the file path is not absolute—that is, beginning with a `/`—it is assumed that the path is relative to the `ServerRoot`. Using the `AuthGroupFile` directive, you can specify a group file in the same manner.

Next, use the `AuthType` directive to set the type of authentication to be used for this resource. Here, the type is set to `Basic`.

Now you need to decide to which realm the resource will belong. Realms are used to group different resources that will share the same users for authorization. A realm can consist of just about any string. The realm is shown in the Authentication dialog box in the user's web browser. Therefore, you should set the realm string to something informative. The realm is defined with the `AuthName` directive.

Finally, state which type of user is authorized to use the resource. You do this with the `require` directive. The three ways to use this directive are as follows:

- ▶ If you specify `valid-user` as an option, any user in the user file is allowed to access the resource (provided that she also enters the correct password).
- ▶ You can specify a list of users who are allowed access with the `users` option.
- ▶ You can specify a list of groups with the `group` option. Entries in the group list, as well as the user list, are separated by spaces.

Returning to the `server-status` example you saw earlier, instead of letting users access the `server-status` resource based on hostname, you can require the users to be authenticated to access the resource. You can do so with the following entry in the configuration file:

```
<Location /server-status>
    SetHandler server-status
    AuthType Basic
```

```

AuthName "Server status"
AuthUserFile "gnulixusers"
Require valid-user
</Location>

```

Final Words on Access Control

If you have host-based as well as user-based access protection on a resource, the default behavior of Apache is to require the requester to satisfy both controls. But say that you want to mix host-based and user-based protection and allow access to a resource if either method succeeds. You can do so by using the `satisfy` directive. You can set the `satisfy` directive to `All` (which is the default) or `Any`. When set to `All`, all access control methods must be satisfied before the resource is served. If `satisfy` is set to `Any`, the resource is served if any access condition is met.

Here's another access control example, again using the previous `server-status` example but this time combining access methods so all users from the `Gnulix` domain are allowed access and those from outside the domain must identify themselves before gaining access:

```

<Location /server-status>
  SetHandler server-status
  Order deny,allow
  Deny from all
  Allow from gnulix.org
  AuthType Basic
  AuthName "Server status"
  AuthUserFile "gnulixusers"
  Require valid-user
  Satisfy Any
</Location>

```

There are more ways to protect material on your web server, but the methods discussed here should get you started and will probably be more than adequate for most circumstances. Look to Apache's online documentation for more examples of how to secure areas of your site.

Apache Modules

The Apache core does relatively little; Apache gains its functionality from modules. Each module solves a well-defined problem by adding necessary features. By adding or removing modules to supply the functionality you want Apache to have, you can tailor the Apache server to suit your exact needs.

A number of core modules are included with the basic Apache server. Many more are available from other developers. The Apache Module Registry is a repository for add-on modules for Apache; you can find it at <https://httpd.apache.org/modules/>. The modules are stored in the `/usr/lib/apache2/modules` directory.

Each module adds new directives that you can use in your configuration files. As you might guess, there are far too many extra commands, switches, and options to describe them all in this chapter. The following sections briefly describe a subset of those modules available with Ubuntu's Apache installation.

To enable a module, use this command:

```
matthew@seymour:~$ sudo a2enmod module_name
```

To disable a module, use this:

```
matthew@seymour:~$ sudo a2dismod module_name
```

Note that you need to use the actual name of the module, not the filename, with both of these commands; for example, `mod_version.so` is the filename, but `version` is the name of the module. You have to know the name of the module to use either command, but in most cases, it is as simple as the difference in this example. Also, after you run either command, you need to restart `apache2` to activate the new configuration.

mod_access

`mod_access` controls access to areas on your web server based on IP addresses, hostnames, or environment variables. For example, you might want to allow anyone from within your own domain to access certain areas of your website. See the “File System Authentication and Access Control” section, earlier in this chapter, for more information.

mod_alias

`mod_alias` manipulates the URLs of incoming HTTP requests, such as redirecting a client request to another URL. It also can map a part of the file system into your web hierarchy. For example, the following fetches contents from the `/home/wsb/graphics` directory for any URL that starts with `/images/`:

```
Alias /images/ /home/wsb/graphics/
```

This is done without the client's knowing anything about it. If you use a redirection, the client is instructed to go to another URL to find the requested content. You can accomplish more advanced URL manipulation with `mod_rewrite`, discussed later in this chapter.

mod_asis

`mod_asis` is used to specify, in fine detail, all the information to be included in a response. This completely bypasses any headers Apache might have otherwise added to the response. All files with an `.asis` extension are sent straight to the client, without any changes.

As a short example of the use of `mod_asis`, assume that you've moved content from one location to another on your site. Now you must inform people who try to access this resource that it has moved, and you also need to automatically redirect them to the new

location. To provide this information and redirection, you can add the following code to a file with the extension `.asis`:

```
Status: 301 No more old stuff!
Location: http://gnulix.org/newstuff/
Content-type: text/html

<HTML>
  <HEAD>
    <TITLE>We've moved...</TITLE>
  </HEAD>
  <BODY>
    <P>We've moved the old stuff and now you'll find it at:</P>
    <A HREF="http://gnulix.org/newstuff/">New stuff</A>!.
  </BODY>
</HTML>
```

mod_auth

`mod_auth` uses a simple user authentication scheme, referred to as Basic Authentication, which is based on storing usernames and encrypted passwords in a text file. This file looks very much like UNIX's `/etc/passwd` file and is created with the `htpasswd` command. See the “File System Authentication and Access Control” section, earlier in this chapter, for more information about this subject.

mod_auth_anon

The `mod_auth_anon` module provides anonymous authentication similar to that of anonymous FTP. The module enables you to define user IDs of those who are to be handled as guest users. When such a user tries to log on, he is prompted to enter his email address as his password. You can have Apache check the password to ensure that it's a (more or less) proper email address. Basically, Apache ensures that the password contains an `@` character and at least one `.` character.

mod_auth_dbm

`mod_auth_dbm` uses Berkeley DB files instead of text for user authentication files.

mod_auth_digest

`mod_auth_digest` is an extension of the basic `mod_auth` module. Instead of sending the user information in plain text, `mod_auth_digest` sends it via the *Message Digest 5 (MD5)* authentication process. This authentication scheme is defined in RFC 2617, “HTTP Authentication: Basic and Digest Access Authentication.” Compared to using Basic Authentication, this is a much more secure way of sending user data over the Internet. Unfortunately, not all web browsers support this authentication scheme.

To create password files for use with `mod_auth_dbm`, you must use the `htdigest` utility. It has more or less the same functionality as the `htpasswd` utility. See the man page of `htdigest` for further information.

mod_autoindex

The `mod_autoindex` module dynamically creates a file list for directory indexing. The list is rendered in a user-friendly manner similar to lists provided by FTP's built-in `ls` command.

mod_cgi

`mod_cgi` allows execution of CGI programs on your server. CGI programs are executable files that reside in the `/var/www/cgi-bin` directory and are used to dynamically generate data (usually HTML) for the remote browser when requested.

mod_dir and mod_env

The `mod_dir` module is used to determine which files are returned automatically when a user tries to access a directory. The default is `index.html`. If you have users who create web pages on Windows systems, you should also include `index.htm`, like this:

```
DirectoryIndex index.html index.htm
```

`mod_env` controls how environment variables are passed to CGI and SSI scripts.

mod_expires

`mod_expires` is used to add an expiration date to content on your site by adding an `Expires` header to the HTTP response. Web browsers and cache servers don't cache expired content.

mod_headers

`mod_headers` is used to manipulate the HTTP headers of your server's responses. You can replace, add, merge, or delete headers as you see fit. The module supplies a `Header` directive for this. Ordering of the `Header` directive is important. A `set` followed by an `unset` for the same HTTP header removes the header altogether. You can place `Header` directives almost anywhere within your configuration files. These directives are processed in the following order:

1. Core server
2. Virtual host
3. `<Directory>` and `.htaccess` files
4. `<Location>`
5. `<Files>`

mod_include

`mod_include` enables the use of server-side includes on your server, which were quite popular before PHP took over this part of the market.

mod_info and mod_log_config

`mod_info` provides comprehensive information about your server's configuration. For example, it displays all the installed modules, as well as all the directives used in its configuration files.

`mod_log_config` defines how your log files should look. See the “Logging” section, later in this chapter, for further information about this subject.

mod_mime and mod_mime_magic

The `mod_mime` module tries to determine the MIME types of files by examining their extensions.

The `mod_mime_magic` module tries to determine the MIME types of files by examining portions of their content.

mod_negotiation

Using the `mod_negotiation` module, you can select one of several document versions that best suits the client's capabilities. There are several options to select which criteria to use in the negotiation process. You can, for example, choose among different languages, graphics file formats, and compression methods.

mod_proxy

`mod_proxy` implements proxy and caching capabilities for an Apache server. It can proxy and cache FTP, CONNECT, HTTP/0.9, and HTTP/1.0 requests. This is not an ideal solution for sites that have a large number of users and therefore have high proxy and cache requirements. However, it is more than adequate for a small number of users.

mod_rewrite

`mod_rewrite` is the Swiss Army knife of URL manipulation. It enables you to perform any imaginable manipulation of URLs, using powerful regular expressions. It provides rewrites, redirection, proxying, and so on. There is little that you cannot accomplish using this module.

TIP

If you have Apache installed and running, see <https://localhost/manual/misc/rewrite-guide.html> for a cookbook that gives you an in-depth explanation of the `mod_rewrite` module's capabilities

mod_setenvif

`mod_setenvif` enables manipulation of environment variables. Using small snippets of text-matching code known as *regular expressions*, you can conditionally change the content of environment variables. The order in which `SetEnvIf` directives appear in the configuration files is important. Each `SetEnvIf` directive can reset an earlier `SetEnvIf` directive when used on the same environment variable. Be sure to keep that in mind when using the directives from this module.

mod_speling

`mod_speling` is used to enable correction of minor typos in URLs. If no file matches the requested URL, this module builds a list of the files in the requested directory and extracts the files that are the closest matches. It tries to correct only one spelling mistake.

mod_status

You can use `mod_status` to create a web page that contains a plethora of information about a running Apache server. The page contains information about the internal status as well as statistics about the running Apache processes. This can be a great aid when you're trying to configure your server for maximum performance. It also provides a good indicator of when something's amiss with your Apache server.

mod_ssl

`mod_ssl` provides Secure Sockets Layer (versions 2 and 3) and Transport Layer Security (version 1) support for Apache. At least 30 directives deal with options for encryption and client authorization and can be used with this module. This mod requires that you also install OpenSSL and generate or buy a certificate. (This is covered later in the chapter, in the "HTTPS" section.)

mod_unique_id

`mod_unique_id` generates a unique request identifier for every incoming request. This ID is put into the `UNIQUE_ID` environment variable.

mod_userdir

The `mod_userdir` module enables mapping of a subdirectory in each user's `/home` directory into your web tree. The module provides several ways to accomplish this.

mod_usertrack

`mod_usertrack` is used to generate a cookie for each user session—for example, to track the user's click stream within your web tree. You must enable a custom log that logs this cookie into a log file.

mod_vhost_alias

`mod_vhost_alias` supports dynamically configured mass virtual hosting, which is useful for Internet service providers (ISPs) with many virtual hosts. However, for the average user, Apache's ordinary virtual hosting support should be sufficient.

There are two ways to host virtual hosts on an Apache server. You can have one IP address with multiple CNAMEs, or you can have multiple IP addresses with one name per address. Apache has different sets of directives to handle each of these options. (You learn more about virtual hosting in Apache in the next section of this chapter.)

Again, the available options and features for Apache modules are too numerous to describe completely in this chapter. You can find complete information about the Apache modules in the online documentation for the server included with Ubuntu or at the Apache Project's website.

Virtual Hosting

One of the most popular services to provide with a web server is to host a virtual domain. Also known as a *virtual host*, a virtual domain is a complete website with its own domain name, as if it were a standalone machine, but it's hosted on the same machine as other websites. Apache implements this capability in a simple way, with directives in the `apache2.conf` configuration file.

Apache now can dynamically host virtual servers by using the `mod_vhost_alias` module you read about in the preceding section of the chapter. The module is primarily intended for ISPs and similar large sites that host a large number of virtual sites. This module is for more advanced users and is therefore beyond the scope of this introductory chapter. This section concentrates on the traditional ways of hosting virtual servers.

Address-Based Virtual Hosts

After you've configured your Linux machine with multiple IP addresses, setting up Apache to serve them as different websites is simple. You need only put a `VirtualHost` directive in your `apache2.conf` file for each of the addresses you want to make an independent website:

```
<VirtualHost 212.85.67.67>
  ServerName gnulix.org
  DocumentRoot /home/virtual/gnulix/public_html
  TransferLog /home/virtual/gnulix/logs/access_log
  ErrorLog /home/virtual/gnulix/logs/error_log
</VirtualHost>
```

Use the IP address, rather than the hostname, in the `VirtualHost` tag.

You can specify any configuration directives within the `<VirtualHost>` tags. For example, you might want to set `AllowOverrides` directives differently for virtual hosts than you do for your main server. Any directives that aren't specified default to the settings for the main server.

Name-Based Virtual Hosts

Name-based virtual hosts enable you to run more than one host on the same IP address. You must add the names to your DNS record as CNAMEs of the machine in question.

When an HTTP client (web browser) requests a document from your server, it sends with the request a variable indicating the server name from which it's requesting the document. Based on this variable, the server determines from which of the virtual hosts it should serve content.

Name-based virtual hosts require just one step more than IP address-based virtual hosts. You must first indicate which IP address has the multiple DNS names on it. This is done with the `NameVirtualHost` directive:

```
NameVirtualHost 212.85.67.67
```

You must then have a section for each name on that address and set the configuration for that name. As with IP-based virtual hosts, you need to set only those configurations that must be different for the host. You must set the `ServerName` directive because it is the only thing that distinguishes one host from another:

```
<VirtualHost 212.85.67.67>
  ServerName bugserver.gnulix.org
  ServerAlias bugserver
  DocumentRoot /home/bugserver/htdocs
  ScriptAlias /home/bugserver/cgi-bin
  TransferLog /home/bugserver/logs/access_log
</VirtualHost>
```

```
<VirtualHost 212.85.67.67>
  ServerName pts.gnulix.org
  ServerAlias pts
  DocumentRoot /home/pts/htdocs
  ScriptAlias /home/pts/cgi-bin
  TransferLog /home/pts/logs/access_log
  ErrorLog /home/pts/logs/error_log
</VirtualHost>
```

TIP

If you are hosting websites on an intranet or internal network, users will likely use the shortened name of the machine rather than the FQDN. For example, users might type `https://bugserver/index.html` in their browser location field rather than `https://bugserver.gnulix.org/index.html`. In that case, Apache would not recognize that those two addresses should go to the same virtual host. You could get around this by setting up `VirtualHost` directives for both `bugserver` and `bugserver.gnulix.org`, but the easy way around it is to use the `ServerAlias` directive, which lists all valid aliases for the machine:

```
ServerAlias bugserver
```

For more information about `VirtualHost`, refer to the help system on `http://localhost/_manual`.

Logging

Apache provides logging for just about any web access information you might be interested in. Logging can help with the following:

- ▶ System resource management, by tracking usage
- ▶ Intrusion detection, by documenting bad HTTP requests
- ▶ Diagnostics, by recording errors in processing requests

Two standard log files are generated when you run your Apache server: `access_log` and `error_log`. They are found under the `/var/log/apache2` directory. (Others include the SSL logs `ssl_access_log`, `ssl_error_log`, and `ssl_request_log`.) All logs except for the `error_log` (by default, this is just the `access_log`) are generated in a format specified by the `CustomLog` and `LogFormat` directives, which appear in your `apache2.conf` file.

A new log format can be defined with the `LogFormat` directive:

```
LogFormat "%h %l %u %t \"%r\" %>s %b" common
```

The common log format is a good starting place for creating your own custom log formats. Note that most of the available log analysis tools assume that you're using the common log format or the combined log format, both of which are defined in the default configuration files.

The following variables are available for `LogFormat` statements:

- ▶ `%a`—Remote IP address.
- ▶ `%A`—Local IP address.
- ▶ `%b`—Bytes sent, excluding HTTP headers. This is shown in Apache's *Combined Log Format (CLF)*. For a request without any data content, a `-` is shown instead of `0`.
- ▶ `%B`—Bytes sent, excluding HTTP headers.
- ▶ `%{VARIABLE}e`—The contents of the environment variable.
- ▶ `%f`—The filename of the output log.
- ▶ `%h`—Remote host.
- ▶ `%H`—Request protocol.
- ▶ `%{HEADER}i`—The contents of `HEADER`; header lines in the request sent to the server.
- ▶ `%l`—Remote log name (from `identd`, if supplied).
- ▶ `%m`—Request method.
- ▶ `%{NOTE}n`—The contents of `NOTE` from another module.
- ▶ `%{HEADER}o`—The contents of `HEADER`; header lines in the reply.
- ▶ `%p`—The canonical port of the server serving the request.

- ▶ `%P`—The PID of the child that serviced the request.
- ▶ `%q`—The contents of the query string, prepended with a `?` character. If there's no query string, this evaluates to an empty string.
- ▶ `%r`—The first line of request.
- ▶ `%s`—Status. For requests that were internally redirected, this is the status of the original request (`%>s` for the last).
- ▶ `%t`—The time, in common log time format.
- ▶ `%{format}t`—The time, in the form given by `format`.
- ▶ `%T`—The seconds taken to serve the request.
- ▶ `%u`—The remote user from `auth`; this might be bogus if the return status (`%s`) is 401.
- ▶ `%U`—The URL path requested.
- ▶ `%V`—The server name according to the `UseCanonicalName` directive.
- ▶ `%v`—The canonical `ServerName` of the server serving the request.

You can put a conditional in front of each variable to determine whether the variable is displayed. If the variable isn't displayed, `-` is displayed instead. These conditionals are in the form of a list of numerical return values. For example, `%!401u` displays the value of `REMOTE_USER` unless the return code is 401.

You can then specify the location and format of a log file using the `CustomLog` directive:

```
CustomLog logs/access_log common
```

If it is not specified as an absolute path, the location of the log file is assumed to be relative to the `ServerRoot`.

HTTPS

The `mod_ssl` module listed earlier gives Apache2 the ability to encrypt communications using OpenSSL. This means your website can be accessed using `https://` instead of just `http://`, and all communications to and from the site will be encrypted. The module is included in the main `apache2-common` package, so if you installed that from the Ubuntu repositories when you installed Apache, you don't have to install additional `apache` packages.

Enter the following to enable the `mod_ssl` module:

```
matthew@seymour:~$ sudo a2enmod ssl
```

This includes a default HTTPS configuration file, found in `/etc/apache2/sites-available/default-ssl.conf`. For HTTPS to work, a certificate and a key are required. The default configuration includes a certificate and key generated by the `ssl-cert` package,

and they are adequate for testing. However, for real use, you should either generate a self-signed certificate and key (which is adequate for internal use or for personal sites) or buy a certificate from a CA (which is necessary if you want anyone to trust your site for commercial ventures).

To configure Apache2 for HTTPS using the default configuration for testing, use this command:

```
matthew@seymour:~$ sudo a2enmsite default-ssl
```

After you restart Apache2, you can access web pages on your server by using https://. This is adequate for testing but not for anything else.

Next we look at how to create a self-signed certificate and key, which is a step in the right direction.

To generate a key for the certificate, use this command:

```
matthew@seymour:~$ openssl genrsa -des3 -out server.key 2048
```

This generates a basic key using Triple DES and 2,048-bit encryption. See the man page for `openssl` for more information about possible settings.

To generate a certificate signing request (CSR), use this command:

```
matthew@seymour:~$ openssl req -new -key server.key -out server.csr
```

You are then asked for some information to complete the request.

To generate a self-signed certificate, use this command:

```
matthew@seymour:~$ openssl x509 -req -days 365 -in server.csr -signkey server.key -out server.crt
```

This creates a certificate that is valid for 365 days. Certificates, even from vendors, have expiration dates. Certificates should be renewed regularly to reassure your site visitors that they are dealing with who they think they are dealing with.

To copy the certificate to its proper location, use this command:

```
matthew@seymour:~$ cp server.crt /etc/ssl/certs/
```

To copy the key to its proper location, use this command:

```
matthew@seymour:~$ cp server.key /etc/ssl/private/
```

Next we look at how to edit the file `/etc/apache2/sites-available/default-ssl` to change the values of these lines to what we show here:

```
SSLEngine on
SSLCertificateFile /etc/ssl/certs/server.crt
SSLCertificateKeyFile /etc/ssl/private/server.key
```


This tells Apache2 to use SSL and where to find the proper certificate and key files.

To configure Apache2 for HTTPS using the edited default configuration with the self-signed certificate and key file, use this command:

```
matthew@seymour:~$ sudo a2enmsite default-ssl
```

When you restart Apache2, you are asked to input the certificate's key password. Enter it when requested. You now have a server that is secure and good for internal use but not for a customer-facing production environment.

The best thing to do if you are going to host a professional site is to use a CA. Every CA has a preferred method, and you should read a CA's requirements before you use that CA. The basic process is usually like this:

1. Create a private and public encryption key pair.
2. Create a certificate based on the public key.
3. Create a certificate request with information about your server and the company hosting it.
4. Send your certificate request and public key along with proof of your company's identity and payment to the CA.
5. Wait for the CA to verify the request and your identity and send back a certificate like the self-signed one created earlier but signed by the CA.
6. Install that certificate on your server and configure Apache2 to use it.

A CA-signed certificate provides advantages. First, browsers are built with data about most CAs and automatically recognize a signature from one of them on your certificate most of the time. A self-signed certificate will cause the browser to display a rather scary-looking (to a nontechnical person) warning and require the user to acknowledge it before viewing your site. In addition, when the CA issues the signed certificate, it is guaranteeing the identity of the organization providing the web pages.

To learn more about certificates and keys, including installation of keys and certificates you pay for, see <https://tldp.org/HOWTO/SSL-Certificates-HOWTO/index.html>.

References

- ▶ https://news.netcraft.com/archives/web_server_survey.html—A statistical graph of web server usage by millions of servers, showing that Apache is by far the most widely used server for Internet sites
- ▶ <https://httpd.apache.org>—The Apache HTTP Server Project website, where you can find extensive documentation and information about Apache

CHAPTER 26

Nginx Web Server Management

This chapter covers configuration and management of the Nginx web server and includes an overview of some of the major components of the server and discussions of server configuration. In this chapter, you learn how to start, stop, and restart Nginx using the command line. The chapter begins with some introductory information and then shows you how to install, configure, and use Nginx.

About the Nginx Web Server

Pronounced “engine-x,” Nginx is a lightweight and extremely fast web server. It is free and open source. Some well-known websites, such as GitHub, Netflix, and WordPress.com, use Nginx because it is stable and fast under high-traffic conditions while using few resources. It is not as configurable as Apache, but for specific use cases, it is an excellent option and is quite easy to set up and use.

NEWS

The W3Techs website, which tracks trends on the Web, posted an article in July 2013 titled “Nginx just became the most used web server among the top 1000 websites (see http://w3techs.com/blog/entry/nginx_just_became_the_most_used_web_server_among_the_top_1000_websites.)” The article summary says that “34.9% of the top 1000 web sites rely on Nginx. That makes it the most trusted web server on high traffic sites, just ahead of Apache.”

The original design of Nginx was created to allow higher numbers of concurrent website requests. Larger websites often have tens of thousands of clients connected simultaneously, each one making HTTP requests that must be

IN THIS CHAPTER

- ▶ About the Nginx Web Server
- ▶ Installing the Nginx Server
- ▶ Configuring the Nginx Server
- ▶ Virtual Hosting
- ▶ Setting Up PHP
- ▶ Adding and Configuring Modules
- ▶ HTTPS
- ▶ Reference

responded to. The designers of Nginx heard this problem described as C10K and decided they could write a web server that would be capable of serving at least 10,000 clients simultaneously.

THE C10K PROBLEM

The canonical website for learning more about this problem is www.kegel.com/c10k.html. The article provided at this link is from the early 2000s and describes ideas for configuring operating systems and writing code to solve the problem of serving at least 10,000 simultaneous clients from a web server. Today, this problem is even more common, and with the continuing maturity of Nginx, lighttpd, and other web servers, many of the largest, highest-traffic sites have switched away from Apache.

Newer versions of the Apache web server and other modern web servers rely on the concept of threads. *Threads* are kind of like lightweight processes. This deserves some explanation. A process is a specific instance of a computer program running. The process contains both the machine code (the binary, or the compiled version of the program that the computer processor can understand and obey—which is either precompiled as in C or C++ programs or may be the output of a just-in-time compilation as happens with languages like Python or Perl) as well as the current activity of that program, such as the calculations it is performing or the data it stores in memory on which it is operating. Serving an HTTP page by running a complete process each time would be bad because the server's resources would be quickly used up if the site were even moderately popular. Process after process would be started, and they would fight for attention. A thread is the ordered control of a program: First, do this; then, do that; finally, do this other thing. One process may control many threads. This is good for resource management. By using threads instead of processes, a larger number of client requests can be served using fewer system resources than with a process-based server.

Most web servers have traditionally been either process based or thread based. There are also examples of hybrid models, where many multithread processes are used. Process-based servers are great because they are stable, and the crash of one process does not affect other processes. However, they cannot handle as many clients because the creation and destruction of all of those processes creates a lot of processor overhead and requires a large amount of memory. Thread-based servers are great because requests can share memory, making them more efficient and thereby able to serve more requests more quickly. However, a crash in one thread could bring down the entire server.

What makes Nginx different is that it uses an event-driven architecture to handle requests. Instead of starting a new process or a new thread for each request, in an event-driven architecture, the flow of the program is controlled by events, some form of message sent from another process or thread. Here, you have a process that runs as a “listener” or an “event detector” that waits for a request to come in to the server. When the request arrives, instead of a new process starting, the listener sends a message to a different part of the server, called an *event handler*, which then performs a task. The simplified outcome of serving web pages this way is that less processor time and less memory are needed.

There are some significant difficulties inherent in using this method, not the least of which can be greater code complexity, which can make fixing bugs, adding features, and understanding code as a newcomer (or even a returning veteran of the same codebase who has been away from it for a little while) more difficult.

Nginx is designed to scale well from one small, low-powered server up to large networks involving many servers.

For configuration, Nginx uses a system of virtual hosts, similar to Apache, and a similar set of configuration files. The differences are semantic and not terribly difficult. You can create URL rewrites in Nginx using a rather different syntax. In addition, there is nothing similar to Apache's rewrite conditions. Occasional blog posts and web tutorials give some workarounds to handle some of the common forms, but if this is something you do often, you might not want to use Nginx.

One file that many people use and love in Apache is `.htaccess`. There is nothing similar to that in Nginx, so if you need the ability to make changes to rewrites or configurations without restarting the server, you are out of luck. This is probably the primary reason that you don't see shared hosting offering Nginx.

Finally, another well-documented option is to use both Apache and Nginx together, with Apache handling any and all dynamic requests and Nginx handling all static requests. This is faster and lighter than using Apache alone but comes with the burden of added complexity and the increased potential for problems that complexity brings.

Installing the Nginx Server

You can install Nginx through APT or build it yourself from source code. The Nginx source builds on just about any UNIX-like operating system and on Win32.

If you are about to install a new version of Nginx, shut down the old server. Even if it's unlikely that the old server will interfere with the installation procedure, shutting it down ensures that there will be no problems. If you do not know how to stop Nginx, see the section "Configuring the Nginx Server," later in this chapter.

Installing from the Ubuntu Repositories

You can install the `nginx` package from the Ubuntu software repositories. Updated packages usually contain important bug and security fixes. When an updated version is released, install it as quickly as possible to keep your system secure.

For more information about installing software from the Ubuntu repositories, see Chapter 9, "Managing Software."

Building the Source Yourself

You can download the Nginx source directly from <https://wiki.nginx.org/Install>.

When you have the `tar` file, you must open it in a temporary directory, such as `/tmp`. Opening this `tar` file creates a directory called `nginx_version_number`, where `version_number` is the version you have downloaded (for example, `nginx_1.11.3`).

TIP

As with many other software packages distributed in source code form for Linux and other UNIX-like operating systems, extracting the source code results in a directory that contains a `README` and an `INSTALL` file. Be sure to peruse the `INSTALL` file before attempting to build and install the software.

Using `./configure` to Build Nginx

To build Nginx, run the `./configure` script in the directory just created by using this command:

```
matthew@seymour:~$ ./configure
```

This generates the makefile that is used to compile the server code.

Next, run `make` to compile the server code:

```
matthew@seymour:~$ make
```

When the compilation is complete, install the server. This may only be done using admin privileges:

```
matthew@seymour:~$ sudo make install
```

TIP

Using Ubuntu's version of Nginx until you really know what happens at system startup is strongly recommended. No "uninstall" option is available when you install Nginx from source! For the remainder of this chapter, we assume that you have installed Nginx this way. If you install from source, you should check the Nginx documentation as there may be differences from what we describe here.

Configuring the Nginx Server

You can now configure the server. Nginx is most commonly run using virtual hosts, like what most people do with Apache these days. The process is similar, with mainly syntax differences.

If you install using the package manager, all configuration files for Nginx are located in `/etc/nginx`. The primary configuration file is `/etc/nginx/nginx.conf`. Here is an example of that file:

```
user www-data;
worker_processes 1;

# error_log /var/log/nginx/error.log;

events {
    worker_connections 1024;
}
```

```

http {
    include      /etc/nginx/mime.types;
    default_type application/octet-stream;

    access_log  /var/log/nginx/access.log;

    sendfile    on;

    tcp_nopush  on;
    #tcp_nodelay on;

    #keepalive_timeout 0;
    keepalive_timeout 65;

    gzip on;

    include /etc/nginx/sites-enabled/*;
}

```

The `nginx.conf` file contains these parts:

- ▶ **user**—Sets the system user that will be used to run Nginx. This is `www-data` by default. You can add a group to this setting by inserting a second entry:


```
user www-data;
```
- ▶ **worker processes**—Allows you to set how many processes Nginx may spawn on your server. The default value of 1 is fine for most users, although some recommend setting this as high as 4. You can experiment, but do so carefully.
- ▶ **error_log**—This is commented out in this example. You can set the location for an error log by removing the `#` that marks the line as a comment that should not be processed by Nginx and adjusting the listed directory location, if you don't want to use the default log location.
- ▶ **events** and **worker_connections**—These settings adjust how many concurrent connections Nginx will allow per process. It may be helpful to think of it this way: `worker_connections` times `worker_processes` will give you the maximum number of clients that may connect to your server concurrently. Memory plays a factor in whether your server can actually serve all the permitted connections you configure, so if you aren't sure whether your server has enough memory to go higher, leave both settings at their defaults, and you should be fine.
- ▶ **http**—This section contains the base settings for HTTP access:
 - ▶ Leave the `include` and `default_type` lines alone unless you enjoy trying to figure out why content is not being displayed or you know you must adjust which types of content are permitted.

- ▶ Feel free to adjust the location of the `access_log`, which records all attempts to connect to your server, or comment out the line to disable it.
- ▶ `sendfile` is used when you permit Nginx to ignore the contents of the file it is sending, such as when serving larger files that do not require a multiple-request-and-confirmation system when being served, thereby freeing system resources for items that do require Nginx to watch over. Leaving this setting on unless you know why you are turning it off is recommended because it saves resources when serving things like graphics.
- ▶ `tcp_nopush` sends HTTP response headers in one packet, and this is actually a pretty good thing, but experimenting with it is okay.
- ▶ `tcp_nodelay` is for use with items that do not require a response, but most general web use does demand responses, so this is often best commented out, although trying it to see whether it makes a difference in your circumstance is okay.
- ▶ `keepalive_timeout` sets the number of seconds that Nginx will keep a connection open when a request is made. The default is to keep this connection open for more than a minute, which seems a bit odd because you have a limited number of connections available, and keeping a connection alive prevents another requester from using that slot again until the timeout occurs. Setting this to a low number like 2 or 3 seconds seems to permit more people to connect in a minute than would be able to do so otherwise. If you are serving a website with little traffic, the setting doesn't matter. If you have a lot of traffic, using a lower number is generally a good idea.
- ▶ `gzip` allows the use of on-the-fly gzip compression, which can make data transfers a bit faster.
- ▶ `include` defines files that are located outside the `nginx.conf` file that are to be read by Nginx and used for its configuration. You can include multiple files; just create a new line for each. Our example includes a directive to include everything listed in a directory that is generally used for virtual hosts, as described in the section “Virtual Hosting,” which follows. You could place a `server` variable here and define it (as you will do in the “Virtual Hosts” section), instead of the `include`, but if you have multiple sites, configuring how Nginx works for each site individually can be useful, especially if you have one site that has a lot of traffic and another that has little.

Whenever you make a change to the `nginx.conf` file, you must restart Nginx to reload the configuration into Nginx for the changes to take effect, like this:

```
matthew@seymour:~$ sudo systemctl start nginx
```

Some prefer to stop and start it, perhaps doing the configuration work in between. This is quite common as in the past Nginx had a habit of not performing restarts perfectly. In our experience, bad resets are rare, but using this method prevents any doubt:

```
matthew@seymour:~$ sudo systemctl stop nginx
matthew@seymour:~$ sudo systemctl start nginx
```

Virtual Hosting

One of the most popular services to provide with a web server is to host a virtual domain. Also known as a *virtual host*, a virtual domain is a complete website with its own domain name, as if it were a standalone machine, but it's hosted on the same machine as other websites.

Nginx implements this capability in a simple way: Just create a configuration file for your virtual host, name it for that host, and place it in `/etc/nginx/sites-enabled`. We prefer to place our files in `/etc/nginx/sites-available` and then create a symlink in `sites-enabled`, but that is not a requirement. Doing so does allow you to disable one out of several sites by simply deleting the symlink and reloading the configuration or restarting Nginx while preserving the complete configuration for the site, so we recommend you give it a try.

You can place the files for your website wherever you like. The configuration files for Nginx tell the web server where to find them. We like to create them in the main website creator's/maintainer's directory, but others prefer `/var/www`, and still others opt for a different location. Choose a location you like and make a note of it. You will need it shortly.

Here is an example of a file for a virtual host, which we will call `yourdomain.com`. Name the file `yourdomain.com` and place it in `sites-enabled` or `sites-available`, as described earlier. This file includes comments that will help you fill in your specific details:

```
#this first server module is just a rewrite directive - it is not required, and you
#can make the rewrite go the other way, to force NOT using www
server {
    listen    80;                                #sets the HTTP port from which the website is served
    server_name  www.yourdomain.com; #names the server using the www prefix

    #if a server request is made without www, this next line will rewrite it
    rewrite ^/(.*) http://yourdomain.com/$1 permanent;
}

#this second server module tells Nginx where to find the files when requested
server {
    listen    80;                                #sets the HTTP port from which the website is served
    server_name  yourdomain.com; #names the server being configured
    location / {                                #sets the location of the files being served
        root /home/<yourusername>/public_html/yourdomain.com; #top directory for the site
        index index.html;
    }
}
```


Earlier in this section, we mentioned that you may place the files for your website wherever you like. The root line in the file just created is where you place this information. Here we use `<yourusername>`'s personal home folder and place within it a directory called `public_html` specifically created for holding website files. Because our example anticipates serving multiple websites with this one server, it goes further and creates a directory for the sample website, `yourdomain.com`.

At this point, everything should work for simple HTML sites. To add additional domains, repeat these steps for each domain being served.

Setting Up PHP

If you have CGI or other script content, such as a website written in PHP, like WordPress or Drupal, you have more work to do. While the ability to serve PHP is available by default in Nginx (it didn't use to be), it still requires additional setup. In this section you will add the ability to serve PHP content to an existing Nginx server. As a result, there are some PHP-specific parts to this process; they should be obvious.

First, you need to make sure PHP is installed on the server. There are many ways to do this. For this example, you can use PHP-FPM, which you can learn more about at <https://php-fpm.org>.

Install the following packages: `php7.4-cli`, `php7.4-cgi`, `psmisc`, `spawn-fcgi`, and `php7.4-fpm`.

Edit the file `/etc/php7/fpm/pool.d/www.conf` to make `php-fpm` use a UNIX socket instead of a TCP/IP connection by finding this line:

```
listen = 127.0.0.1:9000
```

and replacing it with this:

```
listen = /tmp/php7-fpm.sock
```

This is where you need to check the requirements of whatever PHP-based application you intend to install and use. Some require other PHP extensions. You may also need a database. We will skip this step as the details differ for each application, but this is when you would likely want to take care of these details, although you can probably make them up later. If all you want to do is serve PHP scripts that you have written or run them on your server, you are probably fine not worrying about this and moving on.

Next, you need to edit either `/etc/nginx/nginx.conf` or the file for your virtual host, like `/etc/nginx/sites-enabled/yourdomain.com` from the earlier example, to include information that Nginx needs to be able to deal with PHP content. Editing either will work; the difference is that editing the virtual host file will affect only that website, whereas editing `nginx.conf` will affect everything on your server.

Here is an example of an edited `server` module in one of these files. Note what is moved and what is added from the previous example:

```
server {
    listen 80; #sets the HTTP port from which the website is served
    server_name www.yourdomain.com; #names the server using the www prefix
```

```

#if a server request is made without www, this next line will rewrite it
rewrite ^/(.*) http://yourdomain.com/$1 permanent;
}

#this second server module tells Nginx where to find the files when requested
server {
    listen 80; #sets the HTTP port from which the website is served
    server_name yourdomain.com; #names the server being configured
    root /home/<yourusername>/public_html/yourdomain.com/; #top directory for the site
    index index.html index.php;
    client_max_body_size 1G;
    fastcgi_buffers 64 4K;
    location / { #sets the location of the files being served
        try_files $uri =404;
        include fastcgi_params;
        fastcgi_split_path_info ^(.+\.(php|/.+)$);
        fastcgi_pass unix:/tmp/php5-fpm.sock;
        fastcgi_param SCRIPT_FILENAME $document_root$fastcgi_script_name;
    }
}

```

Restart Nginx, and you are all set. See <https://wiki.nginx.org/PHPFcgExample> for more details.

Adding and Configuring Modules

The topic of adding and configuring modules is bigger than can or should be covered in this book, where we only intend to introduce you to a wide range of technologies and features available in, with, and for Ubuntu and help you acquire a basic competence with them. In this chapter, we have only set up a basic server. It will serve the needs of most people, but tons of other settings are available for you to use, and many other configuration options exist for core modules and even optional modules you can add to your server.

Adding additional modules is something that must be done when Nginx is compiled, so if you need something that is not included by default in the package from the Ubuntu repositories, you need to read the official documentation, download Nginx from the website, and compile it yourself. The best way to start is to immerse yourself in the official module documentation at <https://nginx.org/en/docs/>.

To configure an enabled module, whether a default module or an optional one, start at the same documentation, find the module you want to configure in the list, and click its entry to learn about the various options. Even the Core module configured in the `nginx.conf` example has a ton of extra options. You may not need any of them, but you may find something useful.

HTTPS

Nginx comes with the ability to encrypt communications using `openssl`. What this means is that your website can be accessed using `https://` instead of just `http://`, and all communications to and from the site will be encrypted.

For HTTPS to work, a certificate and a key are required. You should either generate a self-signed certificate and key (which is adequate for internal use or for personal sites) or buy a certificate from a certificate authority (CA) (which is necessary if you want anyone to trust your site for commercial ventures).

To generate a key for the certificate, use this command:

```
matthew@seymour:~$ openssl genrsa -des3 -out server.key 2048
```

This generates a basic key using Triple DES and 2,048-bit encryption. See the man page for `openssl` for more information about possible settings.

To generate a certificate signing request (CSR), use this command:

```
matthew@seymour:~$ openssl req -new -key server.key -out server.csr
```

You are then asked for some information to complete the request.

To generate a self-signed certificate, use this command:

```
matthew@seymour:~$ openssl x509 -req -days 365 -in server.csr -signkey server.key -out server.crt
```

This creates a certificate that is valid for 365 days. Certificates, even from vendors, have expiration dates. Certificates should be renewed regularly to reassure your site visitors that they are dealing with who they think they are dealing with.

To copy the certificate to its proper location, use this command:

```
matthew@seymour:~$ cp server.crt /etc/nginx/ssl/
```

To copy the key to its proper location, use this command:

```
matthew@seymour:~$ cp server.key /etc/nginx/ssl/
```

Next, you must modify your Nginx configuration to use the server certificate and key files. This is done in the `server` module of the config file. Here is the earlier example, with the additions you need now in **bold**:

```
server {
    listen 80;                                #sets the HTTP port from which the website is served

    listen 443 ssl;
```

```
server_name www.yourdomain.com; #names the server using the www prefix

ssl_certificate /etc/nginx/ssl/server.crt
ssl_certificate /etc/nginx/ssl/server.key

#if a server request is made without www, this next line will rewrite it
rewrite ^/(.*) http://yourdomain.com/$1 permanent;
}
```

You can now access web pages on your server by using `https://`. This is adequate for testing and internal use but not for anything else.

The best thing to do if you are going to host a professional site is to use a CA. Every CA has a preferred method, and you should read a CA's requirements before you use that CA. The basic process is usually like this:

1. Create a private and public encryption key pair.
2. Create a certificate based on the public key.
3. Create a certificate request with information about your server and the company hosting it.
4. Send your certificate request and public key along with proof of your company's identity and payment to the CA.
5. Wait for the CA to verify the request and your identity and send back a certificate like the self-signed one created earlier, but signed by the CA.
6. Install that certificate on your server and configure Apache2 to use it.

A CA-signed certificate provides advantages. First, browsers are built with data about most CAs and automatically recognize a signature from one of them on your certificate most of the time. A self-signed certificate will cause the browser to display a rather scary-looking (to a nontechnical person) warning and require the user to acknowledge it before viewing your site. In addition, when the CA issues the signed certificate, it is guaranteeing the identity of the organization providing the web pages.

To learn more about certificates and keys, including installation of keys and certificates you pay for, see <https://tldp.org/HOWTO/SSL-Certificates-HOWTO/index.html>.

Reference

- <https://nginx.org/en/docs/>—The Nginx wiki website, where you can find extensive documentation and information about Nginx, including installation instructions, downloads, and tips for configuration and use

This page intentionally left blank

CHAPTER 27

Other HTTP Servers

To determine the best web server for your use, consider the needs of the website you manage. Does it need heavy security (for e-commerce), multimedia (music, video, and pictures), or the capability to download files easily? How much are you willing to spend for the software? Do you need software that is easy to maintain and troubleshoot or that includes tech support? The answers to these questions might steer you to something other than what's covered in Chapter 25, "Apache Web Server Management" or Chapter 26, "Nginx Web Server Management." This chapter covers other HTTP servers, including `lighttpd`, Yaws, Cherokee, Jetty, `thttpd`, Apache Tomcat, WildFly, and Caddy.

lighttpd

`lighttpd`, or "lighty," is a speedy, lightweight open source server. Like Nginx, `lighttpd` is designed for high performance and low resource use. YouTube, Wikipedia, and other sites use `lighttpd` for its scalability and quickness. Like Nginx, `lighttpd` uses an event-driven architecture.

One thing that sets `lighttpd` apart from Nginx is that although Nginx natively handles static content incredibly well, it can also require a bit more work to get it functioning as a server for CGI content, such as websites built with PHP. The main WordPress site (which WordPress hosts itself) uses Nginx and is built with PHP, so this is not an insurmountable task with Nginx—and web tutorials abound. With `lighttpd`, the ability to natively function as a server for CGI content is built in, making it quicker to install, configure, and use if this is the sort of site you intend to host.

Configuring `lighttpd` is done using a system of configuration files. The syntax is a bit different from either Apache or Nginx, but to anyone familiar with using either, `lighttpd` will be easy to understand and use.

IN THIS CHAPTER

- ▶ `lighttpd`
- ▶ Yaws
- ▶ Cherokee
- ▶ Jetty
- ▶ `thttpd`
- ▶ Apache Tomcat
- ▶ WildFly
- ▶ Caddy
- ▶ References

Rewrite rules are available for `lighttpd`, and you can do them using conditionals and regular expressions. However, the syntax for doing this is different and takes a little time to study and use effectively.

`lighttpd` has a nice website with professional documentation available. Reading through it should give you a good sense of whether `lighttpd` is suitable for a specific site.

Yaws

Yaws, which stands for *Yet Another Web Server*, is written in the Erlang programming language, which is enough to make it unique and interesting for many people. Erlang is primarily designed for and used to build scalable real-time systems that require high availability. It is often found in telephony applications, instant messaging, commerce, and banking, and it's designed to support concurrency and fault tolerance. Yaws was written from the ground up to be scalable and multithreaded. Like newer versions of Apache, Yaws uses threads—in this case one thread per request—to serve content. What makes Yaws an interesting alternative is how the underlying language deals with concurrent processes. Because it uses Erlang, Yaws should be significantly faster, even using the same base method for serving content.

Yaws is configured using one configuration file. Virtual servers are the standard, and the syntax of the file is familiar enough that most simple configurations should be quick and easy to set up.

Here is where a major weakness comes in: The documentation available from the Yaws website at the time of this writing was quite sparse, and much of it is outdated (and is acknowledged by the site with a disclaimer). This is not unlike earlier experiences with Nginx, which has greatly improved its official documentation; but be warned, unless you have simple needs, you might have to spend a lot of time trying things out, adjusting, and playing to get your site up and running. On the positive side, if you are looking for a fun experiment, perhaps as a student running a server just to learn rather than to host vital content, Yaws could be exactly what you want. Who knows? It is an open source project, and you might find yourself studying Erlang and contributing either code or documentation.

Cherokee

The distinguishing feature of Cherokee among the crowd that claims both speed and lightness is its ease of configuration. Cherokee supports all the big features, like virtual hosts, CGI, load balancing, and so on. It also includes a graphical interface for configuration, which makes it unique among all the options in this chapter. The claim is that you can configure Cherokee without ever editing a configuration file.

Setting up Cherokee is as easy as installing it on your system and then opening the Cherokee admin interface by issuing the `cherokee-admin-launcher` command as root. The downside is that this assumes you are running a graphical user interface on the same system as your web server, which is something most Linux admins do not do. However, fear not. It is possible to use this interface remotely from a system with a web browser and

terminal access via `ssh`, using the remote system's IP address and a specific port (which is configurable).

The other features of Cherokee are comparable to those of `lighttpd` and `Nginx`. If you like what they offer but want to use a GUI for your administration tasks, Cherokee is worth a closer look. The end-user documentation available from the Cherokee site is excellent and should get you up and running easily.

Jetty

If you are hosting a personal WordPress site or a few static HTML pages, Jetty is not what you need. Jetty is an Eclipse Foundation project that is written in Java. The Eclipse Foundation exists to provide open source development tools, software frameworks, and more to anyone who wants to use them. Jetty is a web server *and* client. It is a `javax.servlet` container. It supports Web Sockets and many other integrations. All of Jetty's components are open source and freely available, even for commercial use and distribution. Jetty is used in products like Apache ActiveMQ (<https://activemq.apache.org/>) and Google App Engine (<https://cloud.google.com/appengine/>). It works with Apache Maven (<https://maven.apache.org>) to provide a way to run a web application locally while in development.

Jetty isn't really a standalone web server in the traditional sense. Rather, it creates a way to use Java code for web applications. Jetty is complex enough to warrant a book of its own. If you are developing web applications using Java and want a highly configurable network of connectors and handlers at your disposal (and you know what that phrase means), Jetty is a good choice. Jetty components are simple, plain old Java objects (POJOs). Jetty has an API that makes using it in Java easy. As the website says, Jetty "provides an HTTP server and Servlet container capable of serving static and dynamic content either from a standalone or embedded instantiations." If you need it, it's great. If you don't, it's overkill.

thttpd

`thttpd` is a very light, decidedly not flashy or feature-filled web server. It has not been abandoned, but it does not seem to be updated frequently. It is included here for one main reason: It has an interesting feature that makes it unique. First, we provide a general description of the web server.

`thttpd` is small and simple. It claims to have only slightly more than is necessary to support HTTP 1.1. No bells. No whistles. The positive side is that simple code generally has fewer stability issues, has fewer security and performance problems, and uses less memory. Your virtual hosts can be easily configured in `thttpd` using a familiar format. Getting CGI type content to run is more complicated than with any of the other servers in this chapter as it is not supported out of the box.

Here is the interesting part: `thttpd` has a throttling feature that lets you set maximum byte rates on URLs or URL groups. You can limit the speed or use of your site's bandwidth according to the URL being requested by the client. You could take advantage of this feature, for example, by hosting media files and allowing them to be accessed while ensuring you have plenty of bandwidth available for other users to access your static HTML pages.

thttpd is not updated frequently, so it might not be suitable for use on a production server, although there is some evidence that it was once used on a few well-known websites. It seems that the world has moved on. However, the throttle-by-requested-URL is an interesting idea and makes it worthy of a mention here.

Apache Tomcat

Apache Tomcat is a common and frequently used open source Java servlet container that implements Oracle's Java Servlet and JavaServer Pages (JSP) specifications. By doing so, Tomcat provides the means to run Java code in a web server environment. Java programmers love it because they can write web applications in a language they already know. Tomcat can be used as a standalone pure-Java web server, but it is often used in conjunction with the regular Apache web server or another general-purpose web server. In those instances, Tomcat serves requests from the other web server.

WildFly

WildFly is more than a web server. It is an application runtime for Java web applications. As such, it does more than Tomcat, which may be exactly what you need, or it may be overkill, depending on your project. It is used to host many enterprise-focused applications and is the open source, community-developed upstream from which Red Hat builds its JBoss Enterprise Application Platform (EAP).

Caddy

Caddy is an open source web server written in Go. It uses the Go standard library for its HTTP functionality, with HTTPS enabled by default. It is designed to have fewer moving parts and to run great in containers because it has no dependencies. Caddy manages TLS certificate renewals, Online Certificate Status Protocol (OCSP) stapling, which checks the revocation status of X.509 digital certificates. It can handle static file serving, reverse proxying, and Kubernetes ingress. Caddy is fairly new and worth watching.

References

- ▶ www.lighttpd.net—The main website for lighttpd
- ▶ <https://yaws.hyber.org>—The main website for Yaws
- ▶ www.cherokee-project.com—The main website for Cherokee
- ▶ <https://eclipse.org/jetty/>—The main website for Jetty
- ▶ www.acme.com/software/thttpd/—The main website for thttpd
- ▶ www.wildfly.org—The main website for WildFly
- ▶ <https://caddyserver.com>—The main website for Caddy
- ▶ <http://tomcat.apache.org/>—The main website for Apache Tomcat

CHAPTER 28

Administering Relational Database Services

This chapter is an introduction to MySQL and PostgreSQL, two database systems that are available in the Ubuntu repositories. In this chapter, you learn what these systems do, how the two programs compare, and how to consider their advantages and disadvantages. This information can help you choose and deploy the one that best suits your organization's database needs.

SIMILAR TO MYSQL

In April 2009, Oracle announced it was buying Sun Microsystems, which owned MySQL. The deal was completed in January 2010. Since then, at least two groups have forked the code to ensure that the database stays free and open source while still in active development. These groups have released and are working on MariaDB (<https://mariadb.org>) and Percona Server (www.percona.com/software/percona-server). We suggest keeping an eye on these projects and other potential replacements for MySQL in case Oracle decides that the free MySQL is too much of a competitor to its paid database offerings and kills or weakens the project. It is interesting to note that some pretty big Linux distributions and organizations have already switched away from MySQL, such as Fedora, openSUSE, and Wikipedia (all of which now use MariaDB). The good news is that anything you learn about MySQL from this chapter is likely to apply perfectly to the other options mentioned.

The *database administrator (DBA)* for an organization has several responsibilities, which vary according to the size and operations of the organization, supporting staff, and so on.

IN THIS CHAPTER

- ▶ A Brief Review of Database Basics
- ▶ Choosing a Database: MySQL Versus PostgreSQL
- ▶ Configuring MySQL
- ▶ Configuring PostgreSQL
- ▶ Database Clients
- ▶ References

Depending on the particular organization's structure, if you are the organization's DBA, your responsibilities might include the following:

- ▶ **Installing and maintaining database servers**—You might install and maintain the database software. Maintenance can involve installing patches as well as upgrading the software at the appropriate times. As a DBA, you need to have root access to your system and know how to manage software (see Chapter 9, “Managing Software”). You also need to be aware of kernel, file system, and other security issues.
- ▶ **Installing and maintaining database clients**—The database client is the program used to access the database (you learn more about that later in this chapter, in the section “Database Clients”), either locally or remotely over a network. Your responsibilities might include installing and maintaining these client programs on users' systems. This chapter discusses how to install and work with the clients from both the Linux command line and through its graphical interface database tools.
- ▶ **Managing accounts and users**—Account and user management include adding and deleting users from the database, assigning and administering passwords, and so on. In this chapter, you find out how to grant and revoke user privileges and passwords for MySQL and PostgreSQL.
- ▶ **Ensuring database security**—To ensure database security, you need to be concerned with things like access control, which ensures that only authorized people can access the database, and permissions, which ensure that people who can access the database cannot do things they should not do. In this chapter, you learn how to manage *Secure Shell (SSH)*, web, and local *graphical user interface (GUI)* client access to the database. Planning and overseeing the regular backup of an organization's database and restoring data from those backups is another critical component of securing the database.
- ▶ **Ensuring data integrity**—Of all the information stored on a server's hard disk storage, chances are the information in the database is the most critical. Ensuring data integrity involves planning for multiple-user access and ensuring that changes are not lost or duplicated when more than one user is making changes to the database at the same time.

A Brief Review of Database Basics

Database services under Linux that use the software discussed in this chapter are based on a *client/server* model. Database clients are often used to input data and to query or display query results from the server. You can use the command line or a graphical client to access a running server. Databases generally come in two forms: flat file and relational. A *flat file* database can be as simple as a text file with a space, tab, or some other character delimiting different parts of the information. One example of a simple flat file database is the `/etc/passwd` file. Another example is a simple address book that might look something like this:

```
Doe~John~505 Some Street~Anytown~NY~12345~555-555-1212
```

You can use standard UNIX tools such as `grep`, `awk`, and `perl` to search for and extract information from this primitive database. Although this might work well for a small database such as an address book that only one person uses, flat file databases of this type have several limitations:

- ▶ **They do not scale well**—Flat file databases cannot perform random access on data. They can only perform sequential access. This means they have to scan each line in the file, one by one, to look for specific information. As the size of the database grows, access times increase, and performance decreases.
- ▶ **Flat file databases are unsuitable for multiuser environments**—Depending on how the database is set up, it either enables only one user to access it at a time or allows two users to make changes simultaneously, and the changes could end up overwriting each other, which results in data loss.

These limitations obviously make the flat file database unsuitable for any kind of serious work in even a small business—much less in an enterprise environment. Relational databases, or *relational database management systems (RDBMSs)* to give them their full name, are good at finding the relationships between individual pieces of data. An RDBMS stores data in tables with fields much like those in spreadsheets, making the data searchable and sortable. RDBMSs are the focus of this chapter.

NOSQL

There is an exception to what we just said. A fairly new category of databases is now in use, usually referred to as *NoSQL* in casual conversation. Unlike typical flat file databases, NoSQL databases are a form of structured storage that is suitable for large and high-traffic uses. These databases have been written and put into use in places where flat files are unsuitable but where relational databases are slower than desired. It is important to note that although NoSQL databases work great when scalability and speed are desirable, you cannot be certain with all NoSQL databases that data is replicated and available instantly across a large installation. It generally is, but there is no guarantee of up-to-the-moment data. This is okay for some applications (web search) but would be disastrous for others (like a financial institution).

There are many forms of NoSQL databases, each with different intents and applications. Some are created and used by big names you might recognize, such as Google's BigTable and Apache's Cassandra. Ubuntu uses Apache's CouchDB for several applications. Other common ones include MongoDB and Berkeley DB. In this chapter, we concentrate on the more traditional relational databases that are used for most applications, but we urge you to keep an eye on what is happening in the NoSQL world. For more information about NoSQL databases, see Chapter 29, "NoSQL Databases."

Oracle, DB2, Microsoft SQL Server, and the freely available PostgreSQL and MySQL are all examples of RDBMSs. The following sections discuss how relational databases work and provide a closer look at some of the basic processes involved in administering and using databases. You also learn about SQL, the standard language used to store, retrieve, and manipulate database data.

How Relational Databases Work

An RDBMS stores data in tables, which you can visualize as spreadsheets. Each column in the table is a field; for example, a column might contain a name or an address. Each row in the table is an individual record. The table itself has a name you use to refer to that table when you want to get data out of it or put data into it. Figure 28.1 shows an example of a simple relational database that stores name and address information.

In the example shown in Figure 28.1, the database contains only a single table. Most RDBMS setups are much more complex than this, with a single database containing multiple tables. Figure 28.2 shows an example of a database named `sample_database` that contains two tables.

last_name	first_name	address	city	state	zip	phone
Doe	John	501 Somestreet	Anytown	NY	55011	555-555-1212
Doe	Jane	501 Somestreet	Anytown	NY	55011	555-555-1212
Palmer	John	205 Anystreet	Sometown	NY	55055	123-456-7890
Johnson	Robert	100 Easystreet	Easytown	CT	12345	111-222-3333

FIGURE 28.1 In this visualization of how an RDBMS stores data, the database stores four records (rows) that include name and address information, divided into seven fields (columns) of data.

phonebook		
last_name	first_name	phone
Doe	John	555-555-1212
Doe	Jane	555-555-1212
Palmer	John	555-123-4567
Johnson	Richard	555-111-4321

cd_collection				
id	title	artist	year	rating
1	Mindbomb	The The	1989	4
2	For All You've Done	Hillsong	2004	
3	Trouser Jazz	Mr Scruff	2002	5
4	Natural Elements	Acoustic Alchemy	1988	3
5	Combat Rock	The Clash	1982	4
6	Life for Rent	Dido	2003	5
7	Adiemus 4	Karl Jenkins	2000	4
8	The Two Towers	Howard Shore	2002	5

FIGURE 28.2 A single database can contain two tables—in this case, `phonebook` and `cd_collection`.

In the `sample_database` example, the `phonebook` table contains four records (rows) and each record hold three fields (columns) of data. The `cd_collection` table holds eight records, divided into five fields of data.

If you are thinking that there is no logical relationship between the `phonebook` table and the `cd_collection` table in the `sample_database` example, you are correct. In a relational database, users can store multiple tables of data in a single database, even if the data in one table is unrelated to the data in others.

For example, suppose you run a small company that sells widgets and you have a computerized database of customers. In addition to storing each customer's name, address, and phone number, you want to be able to look up outstanding order and invoice information for any of your customers. You could use three related tables in an RDBMS to store and organize customer data for just those purposes. Figure 28.3 shows an example of such a database.

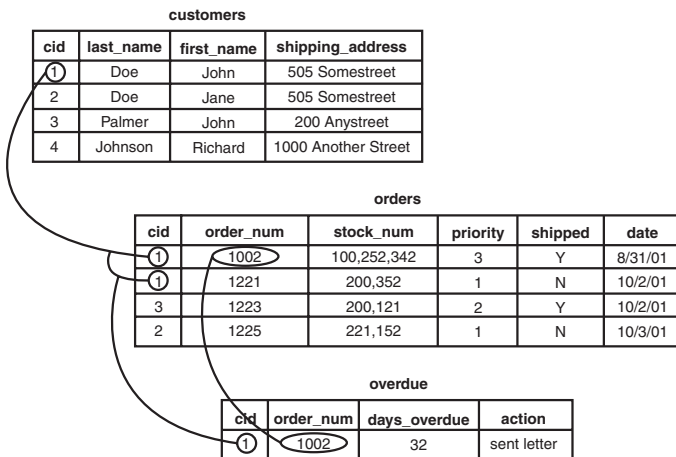


FIGURE 28.3 You can use three related tables to track customers, orders, and outstanding invoices.

In the example in Figure 28.3, we have added a `cid` field to each customer record. This field holds a customer ID number that is the unique piece of information that can be used to link all other information for each customer to track orders and invoices. Each customer is given an ID unique to him; two customers might have the same data in their name fields, but their ID field values will never be the same. The `cid` field data in the `orders` and `overdue` tables replaces the `last_name`, `first_name`, and `shipping_address` field information from the `customers` table. Now, when you want to run a search for any customer's order and invoice data, you can search based on one key rather than multiple keys. You get more accurate results in faster, easier-to-conduct data searches.

Now that you have an idea of how data is stored in an RDBMS and how the RDBMS structure enables you to work with that data, you are ready to learn how to input and output data from the database. This is where SQL comes in.

Understanding SQL Basics

SQL (pronounced “S-Q-L” or “sequel,” depending on who is talking) is a database query language understood by virtually all RDBMSs available today. You use SQL

statements to get data into and retrieve data from a database. As with statements in any language, SQL statements have a defined structure that determines their meanings and functions.

As a DBA, you should understand the basics of SQL, even if you will not be doing any of the actual programming yourself. Fortunately, SQL is similar to standard English, so learning the basics is simple.

Creating Tables

As mentioned previously, an RDBMS stores data in tables that look similar to spreadsheets. Of course, before you can store any data in a database, you need to create the necessary tables and columns to store the data. You do this by using the `CREATE` statement.

For example, the `cd_collection` table from Figure 28.2 has five columns, or fields: `id`, `title`, `artist`, `year`, and `rating`.

SQL provides several column types for data that define what kind of data will be stored in the column. Some of the available types are `INT`, `FLOAT`, `CHAR`, and `VARCHAR`. Both `CHAR` and `VARCHAR` hold text strings, with the difference being that `CHAR` holds a fixed-length string, whereas `VARCHAR` holds a variable-length string.

There are also special column types, such as `DATE`, that only take data in a date format, and `ENUM` (enumeration), which can be used to specify that only certain values are allowed. If, for example, you want to record the genre of your CDs, you could use an `ENUM` column that accepts only the values `POP`, `ROCK`, `EASY_LISTENING`, and so on. You learn more about `ENUM` later in this chapter.

Looking at the `cd_collection` table, you can see that three of the columns hold numeric data and the other two hold string data. In addition, the character strings are of variable length. Based on this information, you can discern that the best type to use for the text columns is type `VARCHAR`, and the best type to use for the others is `INT`. You should notice something else about the `cd_collection` table: One of the CDs is missing a rating, perhaps because we have not listened to it yet. This value, therefore, is optional; it starts empty and can be filled in later.

You are now ready to create a table. As mentioned earlier, you do this by using the `CREATE` statement, which uses the following syntax:

```
CREATE TABLE table_name (column_name column_type(parameters) options, ...);
```

You should know the following about the `CREATE` statement:

- ▶ **SQL commands are not case sensitive**—For example, `CREATE TABLE`, `create table`, and `Create Table` are all valid.
- ▶ **White space is generally ignored**—This means you should use it to make your SQL commands clearer.

The following example shows how to create the table for the `cd_collection` database:

```
CREATE TABLE cd_collection
(
  id INT NOT NULL,
  title VARCHAR(50) NOT NULL,
  artist VARCHAR(50) NOT NULL,
  year VARCHAR(50) NOT NULL,
  rating VARCHAR(50) NULL
);
```

Notice that the statement terminates with a semicolon (;). This is how SQL knows you are finished with all the entries in the statement. In some cases, you can omit the semicolon, and we point out these cases when they arise.

TIP

SQL has a number of reserved keywords that cannot be used in table names or field names. For example, if you keep track of CDs that you want to take with you on vacation, you cannot use the field name `select` because that is a reserved keyword. Instead, you should either choose a different name (perhaps `selected`) or just prefix the field name with an `f`, such as `fselect`.

Inserting Data into Tables

After you create the tables, you can put data into them. You can insert data manually with the `INSERT` statement, which uses the following syntax:

```
INSERT INTO table_name VALUES('value1', 'value2', 'value3', ...);
```

This statement inserts `value1`, `value2`, and so on into the table `table_name`. The values that are inserted constitute one row, or *record*, in the database. Unless specified otherwise, values are inserted in the order in which the columns are listed in the database table. If, for some reason, you want to insert values in a different order (or if you want to insert only a few values and they are not in sequential order), you can specify which columns you want the data to go in by using the following syntax:

```
INSERT INTO table_name (column1, column4) VALUES('value1', 'value2');
```

You can also fill multiple rows with a single `INSERT` statement, using syntax such as the following:

```
INSERT INTO table_name VALUES('value1', 'value2'),('value3', 'value4');
```

In this statement, `value1` and `value2` are inserted into the first row, and `value3` and `value4` are inserted into the second row.

The following example shows how you insert the `Nevermind` entry into the `cd_collection` table:

```
INSERT INTO cd_collection VALUES(9, 'Nevermind', 'Nirvana', '1991', NULL);
```

MySQL requires the `NULL` value for the last column (rating) if you do not want to include a rating. PostgreSQL, in contrast, lets you get away with just omitting the last column. Of course, if you had columns in the middle that were null, you would need to explicitly state `NULL` in the `INSERT` statement.

Normally, `INSERT` statements are coded into a front-end program, so users adding data to the database do not have to worry about the SQL statements involved.

Retrieving Data from a Database

Of course, the main reason for storing data in a database is so you can later look up, sort, and generate reports on that data. Basic data retrieval is done with the `SELECT` statement, which has the following syntax:

```
SELECT column1, column2, column3 FROM table_name WHERE search_criteria;
```

The first two parts of the statement—the `SELECT` and `FROM` parts—are required. The `WHERE` portion of the statement is optional. If it is omitted, all rows in the table `table_name` are returned.

The `column1, column2, column3` syntax allows you to specify the names of the columns you want to see. If you want to see all columns, you can also use the wildcard `*` to show all the columns that match the search criteria. For example, the following statement displays all columns from the `cd_collection` table:

```
SELECT * FROM cd_collection;
```

If you want to see only the titles of all the CDs in the table, you use a statement such as the following:

```
SELECT title FROM cd_collection;
```

To select the title and year of all the CDs in the table, you use the following:

```
SELECT title, year FROM cd_collection;
```

If you want something a little fancier, you can use SQL to print the CD title followed by the year in parentheses, as is the convention. Both MySQL and PostgreSQL provide string concatenation functions to handle problems such as this. However, the syntax is different in the two systems.

In MySQL, you can use the `CONCAT()` function to combine the `title` and `year` columns into one output column, along with parentheses. The following statement is an example:

```
SELECT CONCAT(title,"(",year, ")") AS TitleYear FROM cd_collection;
```

That statement lists both the title and year under one column that has the label `TitleYear`. Note that there are two strings in the `CONCAT()` function along with the fields; these add white space and the parentheses.

In PostgreSQL, the string concatenation function is simply a double pipe (`||`). The following command is the PostgreSQL equivalent of the preceding MySQL command:

```
SELECT (genus||' ('||species||')') AS TitleYear FROM cd_collection;
```

Note that the parentheses are optional, but they make the statement easier to read. Once again, the strings in the middle and at the end (note the space between the quotes) are used to insert spacing and parentheses between the title and year.

Of course, more often than not, you do not want a list of every single row in a database. Rather, you want to find rows that match certain characteristics. For this, you add the `WHERE` statement to the `SELECT` statement. For example, suppose you want to find all the CDs in the `cd_collection` table that have a rating of 5. You would use a statement like the following:

```
SELECT * FROM cd_collection WHERE rating = 5;
```

Using the table from Figure 28.2, you can see that this query would return the rows for *Trouser Jazz*, *Life for Rent*, and *The Two Towers*. This is a simple query, and SQL is capable of handling queries much more complex than this. You can write complex queries using logical `AND` and logical `OR` statements. For example, to refine the query so it lists only CDs that were not released in 2003, you use a query like the following:

```
SELECT * FROM cd_collection WHERE rating = 5 AND year != 2003;
```

In SQL, `!=` means “is not equal to.” So, once again looking at the table from Figure 28.2, you can see that this query returns the rows for *Trouser Jazz* and *The Two Towers* but does not return the row for *Life for Rent* because it was released in 2003.

So, what if you want to list all the CDs that have a rating of 3 or 4 except those released in the year 2000? This time, you combine logical `AND` and logical `OR` statements:

```
SELECT * FROM cd_collection WHERE rating = 3 OR rating = 4 AND year != 2000;
```

This query returns entries for *Mind Bomb*, *Natural Elements*, and *Combat Rock*. However, it does not return entries for *Adiemus 4* because it was released in 2000.

TIP

One of the most common errors among new database programmers is confusing logical `AND` and logical `OR`. For example, in everyday speech, you might say, “Find me all CDs released in 2003 and 2004.” At first glance, you might think that if you fed this statement to the database in SQL format, it would return the rows for *For All You’ve Done* and *Life for Rent*. In fact, it would return no rows at all. This is because the database interprets the statement as “Find all rows in which the CD was released in 2003 and was released in 2004.” It is, of course, impossible for the same CD to be released twice without requiring

a new ISBN and therefore a new database entry, so this statement would never return any rows, no matter how many CDs were stored in the table. The correct way to form this statement is with an `OR` statement instead of an `AND` statement.

SQL is capable of far more than is demonstrated here. But as mentioned earlier, this section is not intended to teach you all there is to know about SQL programming; rather, it teaches you the basics so you can be a more effective DBA.

Choosing a Database: MySQL Versus PostgreSQL

If you are just starting out and learning about using a database with Linux, the first logical step is to research which database will best serve your needs. Many database software packages are available for Linux; some are free, and others cost hundreds of thousands of dollars. Expensive commercial databases, such as Oracle, are beyond the scope of this book. Instead, this chapter focuses on two freely available databases: MySQL and PostgreSQL.

Both of these databases are quite capable, and either one could probably serve your needs. However, each database has a unique set of features and capabilities that might serve your needs better or make developing database applications easier for you.

Speed

Until recently, the speed choice was simple: If the speed of performing queries was paramount to your application, you used MySQL. MySQL has a reputation for being an extremely fast database. Until recently, PostgreSQL was quite slow by comparison.

Newer versions of PostgreSQL have improved in terms of speed (when it comes to disk access, sorting, and so on). In certain situations, such as periods of heavy simultaneous access, PostgreSQL can be significantly faster than MySQL, as you will see in the next section. However, MySQL is still plenty fast compared to many other databases.

Data Locking

To prevent data corruption, a database needs to put a lock on data while it is being accessed. As long as the lock is on, no other process can access the data until the first process has released the lock. This means that any other processes trying to access the data have to wait until the current process completes. The next process in line then locks the data until it is finished, and the remaining processes have to wait their turn, and so on.

Of course, operations on a database generally complete quickly, so in environments with a small number of users simultaneously accessing the database, the locks are usually of such short duration that they do not cause any significant delays. However, in environments in which many people are accessing the database simultaneously, locking can create performance problems as people wait their turn to access the database.

Older versions of MySQL lock data at the table level, which can be a bottleneck for updates during periods of heavy access. This means that when someone writes a row of

data in the table, the entire table is locked so no one else can enter data. If your table has 500,000 rows (or records) in it, all 500,000 rows are locked any time one row is accessed. Once again, in environments with a relatively small number of simultaneous users, this doesn't cause serious performance problems because most operations complete so quickly that the lock time is extremely short. However, in environments in which many people are accessing the data simultaneously, MySQL's table-level locking can be a significant performance bottleneck.

PostgreSQL, in contrast, locks data at the row level. In PostgreSQL, only the row currently being accessed is locked. Other users can access the rest of the table. This row-level locking significantly reduces the performance effect of locking in environments that have a large number of simultaneous users. Therefore, as a general rule, PostgreSQL is better suited for high-load environments than MySQL.

The MySQL release bundled with Ubuntu gives you the choice of using tables with table-level or row-level locking. In MySQL terminology, MyISAM tables use table-level locking and InnoDB tables use row-level locking.

NOTE

MySQL's data locking methods are discussed in more depth at <https://dev.mysql.com/doc/>. You can find more information on PostgreSQL's locking at www.postgresql.org/docs/9.1/static/sql-lock.html.

ACID Compliance in Transaction Processing to Protect Data Integrity

Another way MySQL and PostgreSQL differ is in the amount of protection they provide for keeping data from becoming corrupted. The acronym *ACID* is commonly used to describe several aspects of data protection:

- ▶ **Atomicity**—This means that several database operations are treated as an indivisible (atomic) unit, often called a *transaction*. In a transaction, either all unit operations are carried out or none of them are. In other words, if any operation in the atomic unit fails, the entire atomic unit is canceled.
- ▶ **Consistency**—This aspect ensures that no transaction can cause the database to be left in an inconsistent state. Inconsistent states can be caused by database client crashes, network failures, and similar situations. Consistency ensures that, in such a situation, any transaction or partially completed transaction that would cause the database to be left in an inconsistent state is *rolled back*, or undone.
- ▶ **Isolation**—This ensures that multiple transactions operating on the same data are completely isolated from each other. This prevents data corruption if two users try to write to the same record at the same time. The way isolation is handled can generally be configured by the database programmer. One way that isolation can be handled is through locking, as discussed previously.

- **Durability**—This ensures that, after a transaction has been committed to the database, it cannot be lost in the event of a system crash, network failure, or other problem. This is usually accomplished through transaction logs. Durability means, for example, that if the server crashes, the database can examine the logs when it comes back up, and it can commit any transactions that were not yet complete into the database.

PostgreSQL is ACID compliant, but again MySQL gives you the choice of using ACID-compliant tables or not. MyISAM tables are not ACID compliant, whereas InnoDB tables are. Note that ACID compliancy is no easy task: All the extra precautions incur performance overhead.

SQL Subqueries

Subqueries enable you to combine several operations into one atomic unit, and they enable those operations to access each other's data. By using SQL subqueries, you can perform some extremely complex operations on a database. In addition, using SQL subqueries eliminates the potential problem of data changing between two operations as a result of another user performing some operation on the same set of data. Both PostgreSQL and MySQL have support for subqueries in this release of Ubuntu, but this was not true in earlier releases.

Procedural Languages and Triggers

A *procedural language* is an external programming language that you can use to write functions and procedures. With a procedural language, you can do things that aren't supported by simple SQL. A *trigger* enables you to define an event that invokes the external function or procedure you have written. For example, you can use a trigger to cause an exception if an `INSERT` statement containing an unexpected or out-of-range value for a column is given.

For example, in the CD tracking database, you could use a trigger to cause an exception if a user enters data that does not make sense. PostgreSQL has a procedural language called PL/pgSQL. Although MySQL has support for a limited number of built-in procedures and triggers, it does not have any procedural language. It does have a feature called *stored procedures* that is similar, but it doesn't do quite the same thing.

Configuring MySQL

A free and stable version of MySQL is included with Ubuntu. MySQL is also available from www.mysql.com. The software is available in source code, binary, and APT format for Linux. (See Chapter 9 for the details on adding (or removing) software.)

After you have MySQL installed, you need to initialize the grant tables, which contain information about user accounts, or permissions to access any or all databases and tables and column data within a database. You can do this by issuing `mysql_install_db` as root. This command initializes the grant tables and creates a MySQL root user.

CAUTION

The MySQL data directory needs to be owned by the user that owns the MySQL process, most likely `mysql`. (You might need to change the directory's owner by using the `chown` command.) In addition, only this user should have any permissions on this directory. (In other words, you should set the permissions to `700` by using `chmod`.) Setting up the data directory any other way creates a security hole.

Running `mysql_install_db` should generate output similar to the following:

```
matthew@seymour:~$ sudo mysql_install_db
Preparing db table
Preparing host table
Preparing user table
Preparing func table
Preparing tables_priv table
Preparing columns_priv table
Installing all prepared tables
020916 17:39:05 /usr/libexec/mysqld: Shutdown Complete
...
```

This command prepares MySQL for use on the system and reports helpful information. The next step is to set the password for the MySQL root user, which is discussed in the following section.

CAUTION

By default, the MySQL root user is created with no password. This is one of the first things you must change because the MySQL root user has access to all aspects of the database. The following section explains how to change the password of the user.

Setting a Password for the MySQL Root User

To set a password for the root MySQL user, you need to connect to the MySQL server as the root MySQL user; you can use the command `mysql -u root` to do so. This command connects you to the server with the MySQL client. When you have the MySQL command prompt, issue a command like the following to set a password for the root user:

```
mysql> SET PASSWORD FOR root = PASSWORD("secretword");
```

In this command, replace `secretword` with whatever you want to be the password for the root user. You can use this same command with other usernames to set or change passwords for other database users.

After you enter a password, you can exit the MySQL client by typing `exit` at the command prompt.

Creating a Database in MySQL

In MySQL, you create a database by using the `CREATE DATABASE` statement. To create a database, you connect to the server by typing `mysql -u root -p` and pressing Enter. After you do so, you are connected to the database as the MySQL root user and prompted for a password. After you enter the password, you are placed at the MySQL command prompt. Then you use the `CREATE DATABASE` command. For example, the following commands create a database called `animals`:

```
matthew@seymour:~$ mysql -u root -p
Enter password:
Welcome to the MySQL monitor. Commands end with ; or \g.
Your MySQL connection id is 1 to server version: 3.23.58

Type 'help;' or '\h' for help. Type '\c' to clear the buffer.

mysql> CREATE DATABASE animals;
Query OK, 1 row affected (0.00 sec)
mysql>
```

Another way to create a database is to use the `mysqladmin` command, as the root user, with the `create` keyword and the name of a new database. For example, to create a new database named `reptiles`, you use a command line like this:

```
matthew@seymour:~$ sudo mysqladmin -u root -p create reptiles
Granting and Revoking Privileges in MySQL
```

You probably want to grant yourself some privileges, and eventually you will probably want to grant privileges to other users. Privileges, also known as *rights*, are granted and revoked on four levels:

- ▶ **Global level**—These rights allow access to any database on a server.
- ▶ **Database level**—These rights allow access to all tables in a database.
- ▶ **Table level**—These rights allow access to all columns within a table in a database.
- ▶ **Column level**—These rights allow access to a single column within a database's table.

NOTE

Listing all the available privileges is beyond the scope of this chapter. See the MySQL documentation for more information.

To add a user account, you connect to the database by typing `mysql -u root -p` and pressing Enter. You are then connected as the root user and prompted for a password. (You did set a password for the root user, as instructed in the last section, right?) After you enter the root password, you are placed at the MySQL command prompt.

To grant privileges to a user, you use the `GRANT` statement, which has the following syntax:

```
GRANT what_to_grant ON where_to_grant TO user_name IDENTIFIED BY 'password';
```

The first option, *what_to_grant*, is the privileges you are granting to the user. You specify these privileges with keywords. For example, the `ALL` keyword is used to grant global-, database-, table-, and column-level rights for a specified user.

The second option, *where_to_grant*, specifies the resources on which the privileges should be granted. The third option, *user_name*, is the username to which you want to grant the privileges. Finally, the fourth option, *password*, is a password that should be assigned to this user. If this is an existing user who already has a password and you are modifying permissions, you can omit the `IDENTIFIED BY` portion of the statement.

For example, to grant all privileges on a database named `sampledata` to a user named `foobar`, you could use the following command:

```
GRANT ALL ON animals.* TO foobar IDENTIFIED BY 'secretword';
```

The user `foobar` can now connect to the database `sampledata` by using the password `secretword`, and `foobar` has all privileges on the database, including the ability to create and destroy tables. For example, the user `foobar` can now log in to the server (by using the current hostname—`shuttle2`, in this example) and access the database like this:

```
matthew@seymour:~$ mysql -h shuttle2 -u foobar -p animals
```

```
Enter password:
```

```
Welcome to the MySQL monitor. Commands end with ; or \g.
```

```
Your MySQL connection id is 43 to server version: 3.23.58
```

```
Type 'help;' or '\h' for help. Type '\c' to clear the buffer.
```

```
mysql>
```

NOTE

See the section “The MySQL Command-Line Client” for additional command-line options.

Later, if you need to revoke privileges from `foobar`, you can use the `REVOKE` statement. For example, the following statement revokes all privileges from the user `foobar`:

```
REVOKE ALL ON animals FROM foobar;
```

Advanced database administration, privileges, and security are complex topics that are beyond the scope of this book. See the “References” section at the end of this chapter for links to online documentation. You can also check out Luke Welling’s and Laura Thompson’s book *PHP and MySQL Web Development*.

Configuring PostgreSQL

If you want to use PostgreSQL but do not want to use the version bundled with Ubuntu, you can find the latest PostgreSQL binary files and source at www.postgresql.org. The PostgreSQL packages are distributed as several files. At a minimum, you want the `postgresql` package. You should see the README file in the FTP directory `ftp://ftp.postgresql.org/pub/` to determine whether you need any other packages.

If you are installing from the Ubuntu package files, a necessary `postgres` user account (that is, an account with the name of the user running the server on your system) is created for you automatically:

```
matthew@seymour:~$ fgrep postgres /etc/passwd
postgres:x:26:26:PostgreSQL Server:/var/lib/postgresql:/bin/bash
```

Otherwise, you need to create a user called `postgres` during the installation. This user should not have login privileges because only root should be able to use `su` to become this user, and no one will ever log in directly as the user. (See Chapter 13, “Managing Users,” for more information on how to add users to an Ubuntu system.) After you have added the user, you can install each of the PostgreSQL packages you downloaded using the standard `dpkg -i` command for a default installation.

Initializing the Data Directory in PostgreSQL

Installation initializes the database and sets the permissions on the data directory to their correct values.

CAUTION

The `initdb` program sets the permissions on the data directory to `700`. To avoid creating a security hole, you should not change these permissions to anything else.

You can start the `postmaster` program with the following command (making sure you are still the user `postgres`):

```
matthew@seymour:~$ postmaster -D /usr/local/pgsql/data &
```

If you have decided to use a directory other than `/usr/local/pgsql/data` as the data directory, you should replace the directory in the `postmaster` command line with whatever directory you are using.

TIP

By default, Ubuntu makes the PostgreSQL data directory `/var/lib/pgsql/data`. This is not a very good place to store the data, however, because most people do not have the necessary space in the `/var` partition for any kind of serious data storage. Note that if you change the data directory to something else (such as `/usr/local/pgsql/data`, as in the examples in this section), you need to edit the PostgreSQL startup file (named `postgres`) located in `/etc/init.d` to reflect the change.

Creating a Database in PostgreSQL

Creating a database in PostgreSQL is straightforward, but it must be performed by a user who has permissions to create databases in PostgreSQL—for example, initially the user named `postgres`. You can then simply issue the following command from the shell prompt (not the PSQL client prompt, but a normal shell prompt):

```
matthew@seymour:~# su - postgres
-bash-2.05b$ createdb database
```

where `database` is the name of the database you want to create.

The `createdb` program is actually a wrapper that makes it easier to create databases without having to log in and use `psql`. However, you can also create databases from within `psql` with the `CREATE DATABASE` statement. Here is an example:

```
CREATE DATABASE database;
```

You need to create at least one database before you can start the `psql` client program. You should create this database while you're logged in as the user `postgres`. To log in as this user, you need to use `su` to become root and then use `su` to become the user `postgres`. To connect to the new database, you start the `psql` client program with the name of the new database as a command-line argument, like this:

```
matthew@seymour:~$ psql sampledata
```

If you don't specify the name of a database when you invoke `psql`, the command attempts to connect to a database that has the same name as the user with which you invoke `psql` (that is, the default database).

Creating Database Users in PostgreSQL

To create a database user, you use `su` to become the user `postgres` from the Linux root account. You can then use the PostgreSQL `createuser` command to quickly create a user who is allowed to access databases or create new database users, as follows:

```
matthew@seymour:~$ createuser sandra
Shall the new user be allowed to create databases? (y/n) y
Shall the new user be allowed to create more new users? (y/n) y
CREATE USER
```

In this example, the new user named `sandra` is created and allowed to create new databases and database users. (Carefully consider who is allowed to create new databases or additional users.)

You can also use the PostgreSQL command-line client to create a new user by typing `psql` along with name of the database and then use the `CREATE USER` command to create a new user. Here is an example:

```
CREATE USER foobar ;
```

CAUTION

PostgreSQL allows you to omit the `WITH PASSWORD` portion of the statement. However, doing so causes the user to be created with no password. This is a security hole, so you should always use the `WITH PASSWORD` option when creating users.

NOTE

When you are finished working in the `psql` command-line client, you can type `\q` to get out of it and return to the shell prompt.

Deleting Database Users in PostgreSQL

To delete a database user, you use the `dropuser` command, along with the user's name, and the user's access is removed from the default database, like this:

```
matthew@seymour:~$ dropuser msmith
DROP USER
```

You can also log in to your database by using `psql` and then use the `DROP USER` command. Here is an example:

```
matthew@seymour:~$ psql demodb
Welcome to psql, the PostgreSQL interactive terminal.
```

```
Type: \copyright for distribution terms
      \h for help with SQL commands
      \? for help on internal slash commands
      \g or terminate with semicolon to execute query
      \q to quit
```

```
demodb=# DROP USER msmith ;
DROP USER
demodb=# \q
```

Granting and Revoking Privileges in PostgreSQL

As in MySQL, granting and revoking privileges in PostgreSQL is done with the `GRANT` and `REVOKE` statements. The syntax is the same as in MySQL except that PostgreSQL doesn't use the `IDENTIFIED BY` portion of the statement because with PostgreSQL, passwords are assigned when you create the user with the `CREATE USER` statement, as discussed previously. Here is the syntax of the `GRANT` statement:

```
GRANT what_to_grant ON where_to_grant TO user_name;
```

The following command, for example, grants all privileges to the user `foobar` on the database `sampledata`:

```
GRANT ALL ON sampledata TO foobar;
```

To revoke privileges, you use the `REVOKE` statement. Here is an example:

```
REVOKE ALL ON sampledata FROM foobar;
```

This command removes all privileges from the user `foobar` on the database `sampledata`.

Advanced administration and user configuration are complex topics. This section cannot begin to cover all the aspects of PostgreSQL administration or of privileges and users. For more information about administering PostgreSQL, see the PostgreSQL documentation or consult a book on PostgreSQL, such as *PostgreSQL* by Korry Douglas.

Database Clients

Both MySQL and PostgreSQL use a client/server system for accessing databases. In the simplest terms, the database server handles the requests that come into the database, and the database client handles getting the requests to the server as well as getting the output from the server to the user.

Users never interact directly with the database server, even if it happens to be located on the same machine they are using. All requests to the database server are handled by a database client, which might or might not be running on the same machine as the database server.

Both MySQL and PostgreSQL have command-line clients. A command-line client is a primitive way of interfacing with a database and generally isn't used by end users. As a DBA, however, you use the command-line client to test new queries interactively without having to write front-end programs for that purpose. In later sections of this chapter, you discover a bit about the MySQL graphical client and the web-based database administration interfaces available for both MySQL and PostgreSQL.

The following sections examine two common methods of accessing a remote database, a method of local access to a database server, and the concept of web access to a database.

NOTE

You should consider access and permission issues when setting up a database. Should users be able to create and destroy databases? Or should they only be able to use existing databases? Will users be able to add records to the database and modify existing records? Or should users be limited to read-only access to the database? And what about the rest of the world? Will the general public need to have any kind of access to your database through the Internet? As a DBA, you must determine the answers to these questions.

SSH Access to a Database

Two types of remote database access scenarios are briefly discussed in this section. In the first scenario, the user directly logs in to the database server through *Secure Shell (SSH)* (to take advantage of the security benefits of encrypted sessions) and then starts a program on the server to access the database. In this case, shown in Figure 28.4, the database client is running on the database server itself.

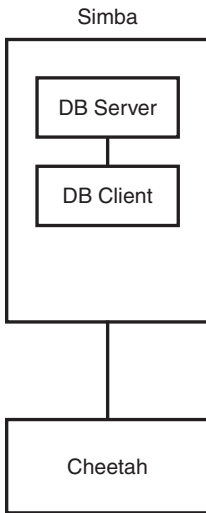


FIGURE 28.4 The user logs in to the database server located on host `simba` from the workstation (host `Cheetah`). The database client is running on `simba`.

In the other scenario, shown in Figure 28.5, the user logs in to a remote host through SSH and starts a program on it to access the database, but the database is actually running on a different system. Three systems are now involved: the user's workstation, the remote host running the database client, and the remote host running the database server.

The important thing to note in Figure 28.5 is the middle system, `Leopard`. Although the client is no longer running on the database server itself, it isn't running on the user's local workstation, either.

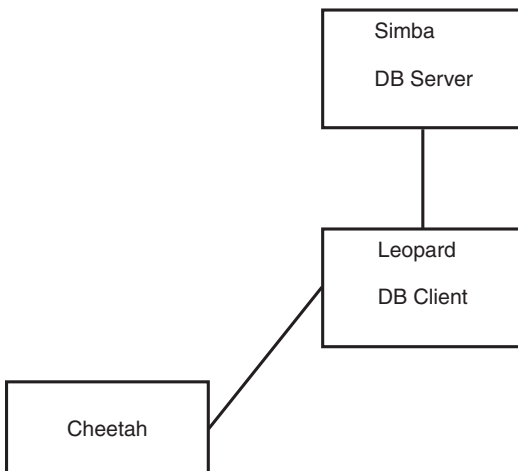


FIGURE 28.5 The user logs in to the remote host, `Leopard`, from the workstation (host `Cheetah`) and starts a database client on `Leopard`. The client on `Leopard` then connects to the database server running on host `simba`. The database client is running on `Leopard`.

Local GUI Client Access to a Database

A user can log in to the database server by using a graphical client (which could be running on Windows, macOS, or a UNIX workstation). The graphical client then connects to the database server. In this case, the client is running on the user's workstation. Figure 28.6 shows an example.

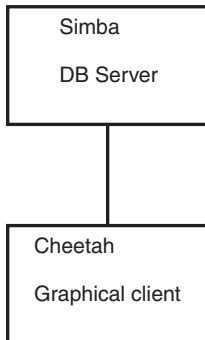


FIGURE 28.6 The user starts a GUI database program on the user's workstation (hostname `Cheetah`). This program, which is the database client, then connects to the database server running on the host `Simba`.

Web Access to a Database

This section looks at two basic examples of web access to the database server. In the first example, a user accesses the database through a form located on the World Wide Web. At first glance, it might appear that the client is running on the user's workstation. Of course, in reality, it is not; the client is actually running on the web server. The web browser on the user's workstation simply provides a way for each user to enter the data that the user wants to send to the database and a way for the results sent from the database to be displayed to the user. The software that actually handles sending the request to the database is running on the web server in the form of a CGI script; a Java servlet; or embedded scripting such as the PHP or Sun Microsystems, Inc.'s JavaServer Pages (JSP).

Often, the terms *client* and *front end* are used interchangeably when speaking of database structures. However, Figure 28.7 shows an example of a form of access in which the client and the front end are not the same thing at all. In this example, the front end is the form displayed in the user's web browser. In such cases, the client is referred to as *middleware*.

In another possible web access scenario, it could be said that the client is a two-piece application in which part of it is running on the user's workstation and the other part is running on the web server. For example, a database programmer can use JavaScript in the web form to ensure that the user has entered a valid query. In this case, the user's query is partially processed on the user's own workstation and partially on the web server. Error checking is done on the user's own workstation, which helps reduce the load on the server and also helps reduce network traffic because the query is checked for errors before being sent across the network to the server.

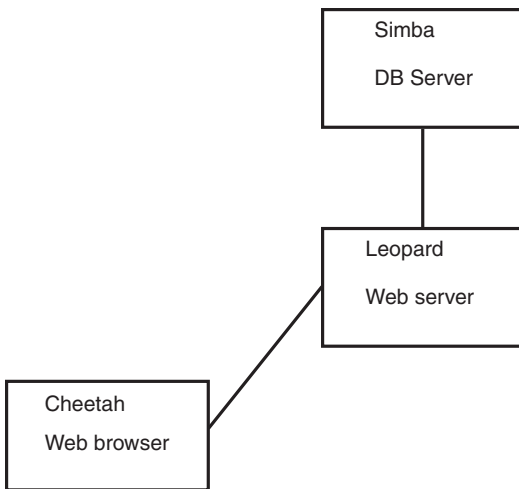


FIGURE 28.7 The user accesses the database through the World Wide Web. The front end is the user's web browser, the client is running on *Leopard*, and the server is running on *Simba*.

The MySQL Command-Line Client

The MySQL command-line client is `mysql`, and it has the following syntax:

```
mysql [options] [database]
```

Some of the available options for `mysql` are discussed in Table 28.1. *database* is optional, and if given, it should be the name of the database to which you want to connect.

Table 28.1 Command-Line Options to Use When Invoking `mysql`

Option	Action
-h <i>hostname</i>	Connects to the remote host <i>hostname</i> (if the database server isn't located on the local system).
-u <i>username</i>	Connects to the database as the user <i>username</i> .
-p	Prompts for a password. This option is required if the user you are connecting as needs a password to access the database. Note that this is a lowercase <i>p</i> .
-P <i>n</i>	Specifies <i>n</i> as the number of the port that the client should connect to. Note that this is an uppercase <i>P</i> .
-?	Displays a help message.

More options are available than are listed in Table 28.1, but these are the most common options. See the man page for `mysql` for more information on the available options.

CAUTION

Although `mysql` allows you to specify the password on the command line after the `-p` option, and thus allows you to avoid having to type the password at the prompt, you should never invoke the client this way. Doing so causes your password to display in the process list, and the process list can be accessed by any user on the system. This is a major security hole, so never give your password on the `mysql` command line.

You can access the MySQL server without specifying a database to use. After you log in, you use the `help` command to get a list of available commands, like this:

```
mysql> help
```

```
MySQL commands:
```

```
Note that all text commands must be first on line and end with ';'.
```

```
help (\h) Display this help.
```

```
? (\?) Synonym for 'help'.
```

```
clear (\c) Clear command.
```

```
connect (\r) Reconnect to the server. Optional arguments are db and host.
```

```
edit (\e) Edit command with $EDITOR.
```

```
ego (\G) Send command to mysql server, display result vertically.
```

```
exit (\q) Exit mysql. Same as quit.
```

```
go (\g) Send command to mysql server.
```

```
nopager (\n) Disable pager, print to stdout.
```

```
notee (\t) Don't write into outfile.
```

```
pager (\P) Set PAGER [to_pager]. Print the query results via PAGER.
```

```
print (\p) Print current command.
```

```
quit (\q) Quit mysql.
```

```
rehash (\#) Rebuild completion hash.
```

```
source (\.) Execute a SQL script file. Takes a file name as an argument.
```

```
status (\s) Get status information from the server.
```

```
tee (\T) Set outfile [to_outfile]. Append everything into given outfile.
```

```
use (\u) Use another database. Takes database name as argument.
```

You can then access a database by using the `use` command and the name of a database that has been created (such as `animals`) and that you are authorized to connect to, as follows:

```
mysql> use animals
```

```
Database changed
```

```
mysql>
```

The PostgreSQL Command-Line Client

You invoke the PostgreSQL command-line client with the command `psql`. As with `mysql`, you can invoke `psql` with the name of the database to which you would like to connect. Also, like `mysql`, `psql` can take several options. These options are listed in Table 28.2.

Table 28.2 Command-Line Options to Use When Invoking `psql`

Option	Action
<code>-h hostname</code>	Connects to the remote host <i>hostname</i> (if the database server isn't located on the local system).
<code>-p n</code>	Specifies <i>n</i> as the number of the port that the client should connect to. Note that this is a lowercase <i>p</i> .
<code>-U username</code>	Connects to the database as the user <i>username</i> .
<code>-W</code>	Prompts for a password after connecting to the database. In PostgreSQL 7 and later, password prompting is automatic if the server requests a password after a connection has been established.
<code>-?</code>	Displays a help message.

Several more options are available in addition to those listed in Table 28.2. See the `psql` man page for details on all the available options.

RELATED UBUNTU AND DATABASE COMMANDS

The following commands are useful for creating and manipulating databases in Ubuntu:

- ▶ `createdb`—Creates a new PostgreSQL database
- ▶ `createuser`—Creates a new PostgreSQL user account
- ▶ `dropdb`—Deletes a PostgreSQL database
- ▶ `dropuser`—Deletes a PostgreSQL user account
- ▶ `mysql`—Interactively queries the MySQL server
- ▶ `mysqladmin`—Administers the MySQL server
- ▶ `mysqldump`—Dumps or backs up MySQL data or tables
- ▶ `psql`—Accesses PostgreSQL via an interactive terminal

Graphical Clients

If you prefer to interact with a database by using a graphical database client rather than with the command-line clients discussed in the previous sections, you are in luck: A few options are available.

MySQL has an official graphical client called *MySQLGUI*. *MySQLGUI* is available in both source and binary formats from the MySQL website at www.mysql.com.

PostgreSQL has *pgAdmin*, which comes with the `postgres` package on Ubuntu.

Web-based administration interfaces are also available for MySQL and PostgreSQL. *phpMyAdmin* and *phpPgAdmin* are two such products. Both of these products are based on the PHP-embedded scripting language and therefore require you to have PHP installed. Of course, you also need to have a web server installed.

References

- ▶ **www.mysql.com**—The official website of the MySQL database server, where you can find the latest versions, up-to-date information, and online documentation for MySQL (You can also purchase support contracts here. You might want to look into this if you will be using MySQL in a corporate setting. Many corporations balk at the idea of using software for which the company has no support contract in place.)
- ▶ **<https://mariadb.org>**—The home page for MariaDB
- ▶ **www.postgresql.org**—The official website of the PostgreSQL database server.

This page intentionally left blank

CHAPTER 29

NoSQL Databases

If you read Chapter 28, “Administering Relational Database Services,” you have already read a brief description of the databases in this chapter. *NoSQL* is a broad term covering a large number of database styles. There are some similarities, but each of these databases was developed for a specific purpose. They are therefore not necessarily interchangeable, although it might be possible to force one to serve a task for which it is not designed. That is rarely a good idea. Also, although there has been a lot of press and hype about NoSQL over the past few years, NoSQL will not and should not be considered a replacement for relational databases. Rather, this is a new set of databases designed to excel in specific situations, especially in large-scale, high-traffic-volume applications. If you have a need for storing and interacting with specific types of data, as described in this chapter, only then do we recommend using one of the databases listed in this chapter.

NOT ONE SIZE FITS ALL

“It’s not a one size fits all anymore. One will use multiple technologies. If I were a CTO, I’d want to use NoSQL for scalable high performance operational data access, lots of reads and writes at high speed and [for] semi real-time and low latency for end users. And you need another for reporting and BI. These [NoSQL] technologies are not optimal for that. In general, a classic data warehouse is a good solution for those things.”

—Dwight Merriman, CEO of MongoDB, at OSCON 2011, as quoted in an article from ZDNet at www.zdnet.com/article/mongodb-chief-it-will-be-mixed-sql-nosql-world/

There are different definitions and even some controversy over what NoSQL means. Does it mean that the database does not use SQL for interactions? Perhaps, but that is not

IN THIS CHAPTER

- ▶ Key/Value Stores
- ▶ Document Stores
- ▶ Wide Column Stores
- ▶ Graph Stores
- ▶ References

absolute. Does it mean, as some now suggest, “not only SQL?” Maybe. That is certainly a broader and more accurate description, although it is also a bit misleading as it seems to include all the relational databases that use SQL. Here is a well-known secret: There is no consistent definition of the term. With that said, here is a reasonably accurate set of features that the databases that are generally labeled NoSQL share:

- ▶ They store structured data (organized in a way that is defined and identifiable).
- ▶ They do not store data relationally (no tables with rows and columns and relationships between tables).

That’s about it.

The advantages to using a NoSQL option instead of a relational database are as follows:

- ▶ NoSQL databases are designed for really large sets of data and can often handle more than any relational database.
- ▶ NoSQL databases are designed to scale as needed. That is, instead of buying a bigger database server to handle increased load as you would with a relational model, you can add additional database hosts easily and spread the database out across them. This is designed to work with commodity hardware and transparently.
- ▶ Commodity hardware is much cheaper than dedicated, professional relational database server hardware, making NoSQL a less expensive option in many cases.
- ▶ Data models with NoSQL are much more relaxed. Some would call this a disadvantage, but it is included in the advantages list because there are some types of data models that may change frequently. In a relational system, data model changes require taking the database offline to modify the structure. Often, NoSQL databases have very little or even no data model restrictions and can allow quick-and-dirty changes.

The disadvantages to using a NoSQL database are these:

- ▶ Support is not as readily available. Most of the NoSQL databases are relatively small open source projects, which is something that should please most readers of this book. However, this also means that, unlike with most enterprise-focused relational databases, there is probably not any enterprise support available for businesses. This will frighten many managers.
- ▶ Most NoSQL projects are fairly new. This also means that they are untested in large enterprises and businesses. It can also mean that it takes longer to set up because of the learning curve; known solutions can be implemented more quickly.
- ▶ NoSQL is not always ACID (*atomicity, consistency, isolation, durability*) compliant. Instead of guaranteeing that every data transaction is instantly and properly recorded, that only one interaction may occur with a piece of data at any given time, and that only the current version is available to the end user, NoSQL databases often work on a system of replication of data across multiple hosts that each get updated eventually. This may happen quickly, but there is no guarantee that at a

given moment you will retrieve the most up-to-date data. This matters when dealing with financial transactions but might not be a concern with web search results where “close enough” might be all that is wanted or necessary.

- ▶ There is no particular advantage to NoSQL unless you have data that is large enough to benefit from it or your project fits neatly into a specific use case.
- ▶ Migrating to NoSQL from traditional relational databases if the need arises isn’t difficult. You know Donald Knuth’s saying that “premature optimization is the root of all evil.” Designing your site or application using known technology and then migrating later if the need arises might be wiser.

One facet that is sometimes described as an advantage and at other times as a disadvantage relates to administration. Relational databases often must be administered by trained staff. The positive side is that qualified *database administrators (DBAs)* are plentiful, if expensive. NoSQL databases are designed to be created and, at least in theory, require little to no further maintenance. Some pundits claim cost savings; others claim that DBAs will still be needed (though for different tasks) and that there are very few people who are trained and available to perform those tasks because of this perspective and the newness of the database style.

NOW, A QUICK WAFFLE ON THE NAME

Google and others seem to like the name “NewSQL” more than “NoSQL.” Some only apply “NewSQL” to a certain group of databases that are somehow different than “NoSQL” databases. While “NoSQL” is much more common to hear, “NewSQL” is the latest buzzword in the database world. Here is one description why:

“‘NewSQL’ is our shorthand for the various new scalable/high performance SQL database vendors. We have previously referred to these products as ‘ScalableSQL’ to differentiate them from the incumbent relational database products. Since this implies horizontal scalability, which is not necessarily a feature of all the products, we adopted the term ‘NewSQL’ in the new report.

“And to clarify, like NoSQL, NewSQL is not to be taken too literally: the new thing about the NewSQL vendors is the vendor, not the SQL.”

—The 451 Group’s Matt Aslett, as recorded at https://blogs.the451group.com/information_management/2011/04/06/what-we-talk-about-when-we-talk-about-newsq/

An interesting development in the NoSQL world is that specifications are being created for a new database query language called UnQL (pronounced “uncle”), which stands for *Unstructured Query Language*. This is being developed as a joint project by two developers: Richard Hipp, the creator of SQLite; and Damien Katz, the creator of CouchDB. They expect more to join them soon. In a nutshell, the language contains some familiar commands, such as `SELECT`, `INSERT`, `UPDATE`, and `DELETE`. However, it is different from SQL because these commands do not work on tables but rather on collections of unordered sets of objects that are described using *JavaScript Object Notation (JSON)*. You can learn more about UnQL from the first product we have seen that uses it at www.unqlite.org.

The sections that follow group databases by similarities in terms of a variety of standout features. Because NoSQL databases are still fairly new, it is unclear which, if any, will become a long-term standard. For that reason, this chapter gives high-level coverage of a larger number of options rather than deep coverage of a couple of primary options.

Key/Value Stores

Key/value stores are the simplest of the NoSQL databases, at least in the sense of interactions. You have a piece of data of any type; this is your value. You give it a name of some sort; this is your key. Any time you need that specific piece of data, you ask for it using the key. Values might be bits of text, binaries, pretty much anything, and the data type does not need to be defined in advance—or perhaps even at all. The database never needs to know what the value object is, just that it is stored using the given key. These databases have no schema. The contents might be vastly different from one another in type, size, domain, and so on. It is the client, the application that uses the database, that is required to know about the value (what it is and the context in which it can be used). The database merely stores it using a key, knows the key/value pair, and serves the value when requested using its key.

Key/value stores are great for things like contents of a website shopping cart, user preference lists, a post in a social media site. Think of things that are not vital, things that might be useful but that will not cause problems if lost. You would not want to use this for credit card information, personal identification, health records, and such. You would want it for high-traffic sites that need to make sure that a local user has quick and accurate access to the information but where the information can take time to replicate to other database nodes or where it might not require replication across nodes at all, or you might use it where there is heavy access to the database itself but where users are not necessarily using the same data concurrently.

Berkeley DB

Berkeley DB was originally created at the University of California, Berkeley, to create a disk hash table that worked better than an existing solution while also helping the university clean up its free UNIX version called BSD by removing code inherited from AT&T. Several years later, Netscape asked the developers to add some desired features to make Berkeley DB more useful to them. This resulted in spinning off Berkeley DB from the university to a company founded for this purpose called Sleepycat Software, which headed development for many years. As of the purchase of Sleepycat Software in 2006, Berkeley DB is now owned by Oracle.

Although it is listed under key/value stores, this is not the only way to interact with a Berkeley DB database. Support also exists for using SQL and Java. Interaction is accomplished using an *application programming interface (API)*. Berkeley DB is very fast and very small. As a result, it can be found running on large-scale systems and embedded within applications and even running on mobile devices.

Berkeley DB is easily the most mature database mentioned in this chapter and is most notable for its use in many well-known software projects, including Subversion, Postfix, and OpenLDAP. It was even included as a data storage backend for MySQL prior to MySQL 5.1.

Cassandra

Cassandra was developed by Facebook for its inbox searching feature. It was released as an open source project when Facebook turned it over to Apache in 2008. Cassandra is a key/value store that runs on a flexible cluster of nodes and is also a wide column store, like HBase, discussed in the “Wide Column Store” section, later in this chapter. Nodes may be added and removed from the cluster. Data is replicated across multiple nodes of the cluster. There is no central node, and access to data exists from any node; if the node receiving the request does not house the specific data requested, it still services the request by retrieving and sending the data. The main goal of Cassandra is fast retrieval of data, with fault tolerance being handled through replication across nodes and speed adjustments via adding additional nodes to create more access points.

One interesting feature is that Cassandra may be tuned to adjust the trade-off between speed of transactions and consistency of data. When data is stored, it is initially stored in memory and gets sent to disk only when specific criteria are met. This makes interaction very quick. In fact, not all data stored in Cassandra is designed to persist over time, and data might not get written to disk at all. This means that not all readers or seekers of data may find a specific piece, but in cases like Facebook’s need to store inbox search data that has only limited time value (such as search results that could be different tomorrow or even 10 minutes from now), this might not matter at all. In these cases, both access speed and convenience are more important.

Cassandra is being used by Facebook, Twitter, Reddit, and many others.

etcd

The open source project behind etcd is CoreOS, which is working in the container world. CoreOS is now owned by Red Hat, which is owned by IBM. (See Chapter 31, “Containers and Ubuntu,” for more about containers.) This key/value store is designed specifically for containerized deployment across a cluster of machines. It is written in Go and is in production use by many large companies, including Cloud Foundry and anyone using Kubernetes.

The focus of etcd is four-fold: simplicity, security, speed, and reliability. It includes a user-facing API and complete access to the source code via GitHub.

Memcached and MemcacheDB

Memcached stores data requested on a system in RAM for a specific period of time to make retrieving that data faster if it is requested again. The time that data persists can be based on a specific setting, memory needs, and other criteria. The goal is to reduce the number of times that data stores must be accessed. Data that is accessed often is held in memory, from where it is much more quickly retrieved. This can alleviate problems such as a page on a blog that has suddenly become popular as a result of the URL being posted on a social networking site. The spike in traffic could be kept manageable because the content of the blog post is being held in memory instead of being requested over and over from the database.

MemcacheDB is an implementation of the Memcached API that uses a key/value format based on Berkeley DB. However, whereas Memcached is designed as a cache solution to speed up data access from memory, MemcacheDB is designed as a persistent storage engine. Because it uses the same API protocol as Memcached, using it is an easy way to add data persistence where caching is already in place with Memcached.

Memcached is used by sites like Twitter, Reddit, YouTube, and Facebook, and it is also supported and often used by websites based on content management systems, such as Drupal and WordPress.

Redis

Initially released in 2009, Redis is intended for applications where performance and flexibility are more important than persistence and absolute data integrity. It is an open source key/value store written in C. Keys can contain strings, hashes, lists, sets, and stored sets. Redis works in RAM for speed, occasionally dumping to disk. Because actions are performed in memory, they are done faster. Operations include appending to a string, incrementing a hash value, pushing to a list, set computations, and more. Redis is also designed so that master/slave replication is easy to set up.

Riak

Riak is a fault-tolerant, distributed database designed for scalability and use in the cloud. It is masterless, meaning there should be no single point of failure. It is designed for speed, simplicity, and stability. Riak is based on a paper by Amazon describing *Dynamo*, which is an internal, proprietary system owned by Amazon. The Riak wiki describes the database in one place as “the most boring database you’ll ever run in production. No sharding required, just horizontal scaling and straightforward capacity planning. The same operational tasks apply to small clusters and large clusters. More machines does not mean more ops.”

Scylla

Scylla is being developed by a fairly young startup called ScyllaDB. The product compares itself with Cassandra, and the company calls it a “drop-in replacement with lightning fast throughput and ultra-low latency.” Although it is being produced by a small company, it has some big-name financial backers. Scylla is not yet well-known; at the time of this writing, we only know of smaller deployments. However, it is being used on servers owned and used by major companies, so we consider it worth mentioning.

Document Stores

Document stores are designed to store data that is already structured in some form of notation, such as JSON or XML. A document store typically focuses on one specific type of notation and is intended to allow entire objects, including arrays and hashes, to be stored and retrieved at once.

Many times document stores are implemented as a layer between an application and a relational database to hold the output of certain types of queries. For example, it might be convenient to aggregate information that is typically requested together—such as a set of user preferences or name and address information—and store it as one object. Requesting and retrieving only one object that is already formatted in an object notation like JSON is faster than making many database queries, and it supplies preformatted data for the client application that can be used to both style output and display specific data at the same time.

Data that is stored and served this way does not have to fit database-specific formatting requirements in a NoSQL database. There are no tables to relate, and data may be larger or smaller and may include more or less information. This is generally called *semistructured data*. Listings 29.1 and 29.2 are quick snippets of two sets of user preferences in JSON—one that includes many user-set preferences and one that includes only one. The client application could be created to assume a set of default preferences that will be used unless specifically overridden by this file.

LISTING 29.1 Sandra's Preferences

```
{ "userpreferences": {
  "displayName": "Don'tHitOnMe",
  "gender": "DoNotDisplay",
  "siteTheme": "Springtime",
  "postsDisplayed": "25",
  "keepLoggedIn": "True"
}
}
```

LISTING 29.2 Matthew's Preferences

```
{ "userpreferences": {
  "siteTheme": "TieDye",
}
}
```

CouchDB

CouchDB began in 2005 as a self-funded personal project of Damien Katz. In 2008, it was given to Apache, where development continues. The goal of CouchDB is to provide a database useful for serving web applications. The emphasis is on scalability and fault tolerance while using commodity hardware. (*Couch* is an acronym for *cluster of unreliable commodity hardware*.) This is not an easy task, but when done successfully, it lowers costs.

CouchDB uses a RESTful HTTP API that is designed from the beginning to be used on and for the Web. All stored items have a unique uniform resource identifier (URI), and full create, read, update, and delete (CRUD) functions are available directly using standard HTTP calls, making CouchDB very easy to integrate into web applications. These calls can be made from a browser or from a command line, using a tool like cURL, which is available on many typical server platforms, including Ubuntu.

A nice feature of CouchDB is that, unlike many NoSQL options, it is designed with the ability to include ACID compliance. This makes it possible to use CouchDB with more consistency-sensitive data.

CouchDB is written in Erlang, which is a language designed for concurrency. This makes CouchDB even better suited for use in a concurrent distributed system. CouchDB is designed to store JSON document objects.

CouchDB is used by several software and web applications, including many Facebook games and applications, internal use at the BBC, and more.

MongoDB

MongoDB is similar to CouchDB in that both are designed as document stores for JSON objects and, like Cassandra, it is designed for replication and high availability. It is created and supported by a company called 10gen and is newer, with its first public release in 2009. A unique feature for this open source database is that the developer offers commercial, enterprise-class support, training, and consulting. This has made adoption of MongoDB much faster than is typical for NoSQL products.

MongoDB supports *sharding*, which means automatically partitioning data across servers for increased performance and scalability. This produces a form of load and data balancing and also offers a way to add nodes simply. Sharding is also intended to support an automatic failover system where node data is replicated, allowing no single point of failure.

In addition, MongoDB includes support for indexing in a manner that is more extensive and powerful than most NoSQL solutions.

MARKETING HYPE OR GREAT DESIGN?

Consider this perspective on MongoDB:

“MongoDB wasn’t designed in a lab. We built MongoDB from our own experiences building large-scale, high-availability, robust systems. We didn’t start from scratch, we really tried to figure out what was broken, and tackle that. So the way I think about MongoDB is that if you take MySQL, and change the data model from relational to document based, you get a lot of great features: embedded docs for speed, manageability, agile development with schema-less databases, easier horizontal scalability because joins aren’t as important. There are lots of things that work great in relational databases: indexes, dynamic queries, and updates, to name a few; and we haven’t changed much there. For example, the way you design your indexes in MongoDB should be exactly the way you do it in MySQL or Oracle; you just have the option of indexing an embedded field.”

—*Eliot Horowitz, 10gen CTO and co-founder*

Obviously, the people behind MongoDB are good at marketing. At the same time, if you listen closely to the crowd, you don’t hear many negative comments about MongoDB, and it has an impressive list of users, including Craigslist, Shutterfly, SourceForge, the *New York Times*, and GitHub. MongoDB has quickly garnered great respect, and its use is constantly spreading.

BaseX

BaseX was started by Christian Grün at the University of Konstanz in 2005 and was subsequently released using a BSD license in 2007. It is a simple, lightweight database that does not support a lot of features but could be just right for specific applications. Rather than using JSON, like CouchDB and MongoDB, BaseX is designed to store document objects in XML. It supports standard XML tools like XPath and XQuery and also includes a lightweight GUI.

BaseX creates indexes and supports W3C recommendations and standards, ACID-safe transactions, large documents, and various APIs like REST/JAX-RX and XML:DB. Although not as sexy or well known as other options in this section, perhaps because of the newness and popularity of JSON over XML, BaseX is respected and used by many universities and enterprises.

Wide Column Stores

Wide column stores are often referred to as big table stores, after one of the best-known examples, Google's BigTable. Typically, a relational database reads data from tables using rows. Data is then sorted to find only those contents of a row that are needed. Wide column stores change the system by reading data from tables in columns, selecting the attributes first before reading in data. This is more efficient for input and output read-only queries. Wide column stores therefore tend to be very efficient for databases that are mostly used for reading stored data, especially from very large data sets.

Wide column stores use something like tables, with a defined schema for each table. Unlike relational databases, wide column stores do not record relationships between tables. These are not relational databases but are more like maps that show where data exists across multiple dimensions. They are designed for scalability and as distributed systems.

Two examples of wide column stores are discussed here. One more, Cassandra, was discussed earlier in this chapter and fits into both this category and the earlier key/value stores category.

BigTable

BigTable is a proprietary Google product that is used only by Google. It is designed to work with Google's MapReduce framework, which was created to process huge data sets across large clusters of computing nodes.

BigTable stores the massive sets of data used by many Google programs, such as Google Reader, My Search History, Google Earth, YouTube, and Gmail. BigTable is not available for use outside Google.

The papers that describe Google's design for both BigTable and MapReduce are listed in the "References" section at the end of this chapter.

HBase

HBase is the database used by Hadoop, Apache Project's free software application for processing huge amounts of data across large clusters of compute nodes in a cluster. Hadoop is modeled in part after the information in Google's MapReduce and Google File System papers. HBase is to BigTable what Hadoop is to MapReduce.

The main feature of HBase is its ability to host very large tables—on the scale of billions of rows across millions of columns. It is designed to host them on commodity hardware. HBase provides a RESTful web service interface that supports many formats and encodings and is optimized for real-time queries.

Numerous companies are using Hadoop, including some very big names like Amazon, eBay, Facebook, IBM, LinkedIn, Yahoo!, and Rackspace.

Graph Stores

Graph stores, or graph databases, literally store data as a graph. This means the data is represented as a series of nodes and indicates how they relate to each other. In the simplest case, a graph with only one node, only the record and its properties need to be recorded. The properties list can be as short as one or as long as a few million (perhaps more).

Rather than allow that awkwardness to grow, most graph databases start creating new nodes sooner, each node having its own properties and also explicit relationships that tie each node to other nodes. It is the relationships that organize the nodes, and the structure is therefore flexible. A graph can look like a list or a map or a tree or something else entirely.

Graph databases are queried using traversals. A traversal begins at a defined starting node and follows through related nodes to answer questions such as "What classes are my friends taking that I am not enrolled in?" or "If server X has a network connection problem, what web services will be disrupted?" In a graph database, an index is just a special type of traversal—usually something commonly done, such as finding specific nodes or relationships according to a property they share.

Graph stores are not terribly common, but they are beloved by those who promote them. There is less differentiation between the options available in this category, at least compared to the differentiation between the other categories of NoSQL databases in this chapter.

Neo4j

Neo4j is the graph store that most people have heard of in the NoSQL world. It has both a free version and a commercial version. Language bindings exist for Java, Python, and Ruby. It is scalable up to graphs of several billion nodes/relationships/properties on a single machine and can be scaled across multiple machines. It can be deployed on a stand-alone server or as a small-footprint database coexisting on the same machine with other software.

OrientDB

OrientDB is a free database released under the Apache 2.0 license. It uses an indexing algorithm called MVRB-Tree, which it claims is significantly faster than other indexing algorithms. You might remember an older relational database called Orient ODBMS. OrientDB is related to that and can be used with a subset of SQL, but it is a complete rewrite using a document/graph database foundation.

HyperGraphDB

HyperGraphDB is another free option that uses the LGPL. It is designed primarily for use with the semantic web, knowledge management, and artificial intelligence projects. In mathematics, a *hypergraph* is an extension to a standard graph that allows an edge to point to more than two nodes. According to the HyperGraphDB website, “HyperGraphDB extends this even further by allowing edges to point to other edges as well and making every node or edge carry an arbitrary value as payload.” HyperGraphDB seems to be focused on the academic side of things. Students might be especially interested in it because HyperGraphDB appears to be trying out some new research ideas.

FlockDB

Twitter uses FlockDB to store social graphs, such as who follows whom, as well as for some secondary indexes. It is free and open source, using the Apache 2.0 license. It is simpler than other graph databases as it seems to try to solve fewer problems and is designed for one primary use. FlockDB is designed for online, low-latency, high-throughput environments such as websites like Twitter; even then, it’s only for storing specific types of data.

References

- ▶ www.oracle.com/database/berkeley-db/db.html—The main Berkeley DB website
- ▶ <https://cassandra.apache.org>—The main website for Cassandra
- ▶ www.scylladb.com—The main website for Scylla
- ▶ www.memcached.org—The main website for Memcached
- ▶ <https://memcachedb.org>—The main website for MemcacheDB
- ▶ <https://coreos.com/etcd/>—The main website for etcd
- ▶ <https://redis.io>—The main website for Redis
- ▶ <https://basho.com/products/>—The main website for Riak
- ▶ <https://couchdb.apache.org>—The main website for CouchDB
- ▶ <https://mongodb.org>—The main website for MongoDB
- ▶ <https://basex.org>—The main website for BaseX

- ▶ <https://research.google.com/archive/bigtable.html>—A paper describing BigTable
- ▶ <https://research.google.com/archive/mapreduce.html>—A paper describing MapReduce
- ▶ <https://hadoop.apache.org>—The main website for Hadoop
- ▶ <https://hbase.apache.org>—The main website for HBase, the Hadoop database
- ▶ <https://research.google.com/archive/gfs.html>—The paper describing Google File System
- ▶ <https://neo4j.com>—The main website for Neo4j
- ▶ <https://orientdb.org>—The main website for OrientDB
- ▶ www.hypergraphdb.org—The main website for HyperGraphDB
- ▶ <https://github.com/twitter/flockdb>—The main website for FlockDB

CHAPTER 30

Virtualization on Ubuntu

IN THIS CHAPTER

- ▶ KVM
- ▶ VirtualBox
- ▶ VMware
- ▶ Xen
- ▶ References

Virtualization is an important topic today, but it isn't a difficult one to understand—at least conceptually. We cover two distinct use cases in this chapter: server virtualization and virtualization on the desktop. Most of the options covered in this chapter work for either use case. In the sections that follow, this chapter points out specific moments that are focused solely on one use case. There are several scenarios, both large and small, that are helpful to illustrate the potential of virtualization and to give the idea some definition.

For starters, imagine a large corporation or business that processes huge amounts of data. That corporation has many dedicated computers to assist with the task. In the past, it might have used mainframes, single computers capable of performing multiple tasks concurrently while dealing with large data sets and multiple, concurrent users. Today, the same company might use a server farm, a network of smaller computers that is extensible and where specific servers in the network can be dedicated to precise tasks.

The problem is that some of these servers do not get used to their capacity. Take, for example, a payroll server that might get extensive use at certain times but might sit nearly idle at other times. That seems like a waste of resources.

What if a systems administrator could pool the resources of all these machines and then dole out those resources as they are needed? She can do so by using virtualization.

At other times, there is a need for servers that do not exist and that will not be needed in the long term. For example, a statistics department might have a one-time need for extra processing power for a big project. It would take a long time to set up a dedicated server, and it is hard to justify the effort for a one-time task.

What if a system administrator could easily create and destroy servers for a specific task, making them appear, completely configured, as needed and then making them disappear when the need no longer exists, freeing up the physical resources for other uses? He can do so by using virtualization.

Networks of physical servers can be created using virtualization, where the physical resources of the servers are pooled together and then passed out as designated by an administrator. It is as if, instead of having 100 servers, each with 4 processors, 32GB of RAM, and 1TB of physical disk storage, you now have one huge resource pool with 400 processors, 3200GB of RAM, and 100TB of disk space. Virtual machines can then use these resources.

A *virtual machine* (VM) is a computer that operates on top of a virtualization layer, often called a *hypervisor*. It isn't real in the sense that it runs on defined, discrete physical resources, but it does all of the same tasks as a "real" computer. The virtualization layer on which the VM runs defines a set of virtual interfaces for the VM, which appear to the VM's operating system as if they were real network cards, memory, hard drives, and so on. In a sense, virtualization fools the guest operating system in the VM into thinking it is running on specific physical equipment that is emulated by the virtualization software, while the virtualization software takes care of the details of interacting with the actual hardware, which may even change without affecting the VM. This is called *hardware emulation* (or sometimes simply *emulation*).

Virtual machines are flexible; their allocated resources may be changed—and in some cases without any downtime. A VM can be created quickly, as needed, and then removed when it is no longer needed, to make the resources it was using once again available to the pool. Servers that are vital but generally use few resources can be created using far fewer resources than required for the physical servers in the pool. Others that are needed for larger tasks might be able to take advantage of the resources of many physical servers in the pool.

It is possible to create a VM and then save its image so that instead of starting with operating system installation each time a VM is created, the VM starts up with a full operating system and installed programs all configured to work together for a desired task. One neat trick is to run a set of servers locally, and then add compute resources from a cloud computing pool such as Amazon's EC2, Ubuntu Enterprise Cloud (using Eucalyptus), Ubuntu Cloud Infrastructure (using OpenStack), or OpenStack to start up VMs on their network, as needed, using them while paying for the time they are running and then deleting them (see Chapter 32, "Ubuntu and Cloud Computing"). This saves a lot of time and money.

What if you run only one machine, and it is your desktop or laptop? Virtualization might be useful to you, as well. Have you ever wanted to test a different operating system than the one you are using? Perhaps you found that you have a need to run a specific program that runs only on Windows, but you have Ubuntu installed on your system. Maybe you want to run the latest development version of Ubuntu to help with testing, but you don't want to use it as your main system operating system. You might want to try out another distribution of Linux or even BSD. In the past, doing these things typically involved partitioning your hard drive and installing both operating systems side by side. That worked, but you couldn't easily move data from one partition to the other, and you couldn't run both operating systems at the same time. Wouldn't it be great if you could run another operating system as a guest on your same machine? You can with virtualization.

There is a trade-off with virtualization, depending on the software used. Some virtualization software runs as an additional layer on top of another operating system. This is great if you want to test something while running on your local desktop machine, but it can add some unwanted and sometimes unacceptable delays when creating a new server. Other virtualization software runs on or near “bare metal,” meaning the virtualization software is either a part of the operating system kernel or runs as the operating system, and there are no other software layers between it and the physical resources being used and managed. This method is faster but not as convenient on the desktop, at least for users who are not as technically advanced.

Virtualization is not new. For example, IBM had useful virtualization running on its mainframes in the 1960s. What has created the recent buzz is that the technology has become available to perform the task on much less-expensive x86 hardware.

In November 2005, Intel released its first processors that supported an extension called VT-x, which allows virtualization software access to the processor and other hardware. Before that, virtualization on any x86 platform was slow because it required difficult software workarounds and massaging to get it working. VT-x is available on many of Intel’s processors but not necessarily all of them as it is one of the ways that Intel differentiates the processors to segment its marketing across various price points. Also, with some motherboards, the extension is not enabled by default but must be enabled in the BIOS before it becomes available.

Following closely behind, in May 2006 AMD released virtualization extensions for its processors. Called AMD-V, these extensions are available on many of AMD’s processors but not all of them, as this is a higher-end feature.

A related topic deserves a quick mention here. In the cloud, the idea of containers has become quite popular. If you are interested in virtualization for use in cloud computing, read the coverage of containers in Chapter 32 before you make any decisions.

KVM

Kernel-based Virtual Machine (KVM) is a part of the Linux kernel. The KVM does not perform hardware emulation but only provides the lower-level tasks. It needs a second layer to run in user space. This is much faster than running the entire virtualization process in user space, on top of another operating system. KVM is designed for use on processors that have either the VT-x or AMD-V extension enabled. Managing VMs with KVM in Ubuntu is accomplished using `libvirt` and `QEMU`. You can check whether a system has the extensions enabled by installing and running the `kvm-ok` package. It is a simple command-line tool that exits with output `0` if the system is suitable or `non-0` if not.

Start by installing the following packages from the Ubuntu software repositories:

- ▶ `qemu-kvm`—The necessary user-space component of KVM
- ▶ `libvirt-bin`—A binary of a C toolkit to interact with the virtualization capabilities of Linux that currently supports not only KVM but also Xen, VirtualBox, and more

- ▶ **virtinst**—A set of command-line tools for creating VMs
- ▶ **bridge-utils**—A set of utilities for configuring Ethernet connections in Linux

You might want to add `virt-viewer`, which provides a nice GUI and VNC interface to VMs, and `virt-manager`, which provides a nice GUI for managing VMs. If installed, you can find both in the Dash listing of applications.

After you install the packages, log out and back in so that the automatic addition of your user to the `libvirt` group is certain to be made effective.

By default, any operating system you install as a guest using KVM has access to network services but is not visible to other machines on the network. It is able to download software updates and browse the Web, for example, but it cannot run as a server accessible by other systems. By default, VMs receive an IP address in the 10.0.2.0/24 range, and hosts are reachable from within a VM by using 10.0.2.2. This should be adequate for simple uses such as testing other operating systems, copying files back and forth using `scp`, or making *virtual private network (VPN)* connections from a host to a guest.

BRIDGED NETWORKING

If you want to change the network settings to enable the use of a VM as an outside-accessible server, you need bridged networking. This enables VMs to use a physical interface to connect to the outside network, making them appear to the rest of the network as any other typical server. Note that to do this you must not use the default Network Manager to control the hardware being bridged. Also, this works only with wired, not wireless, hardware. See Chapter 18, “Networking,” if you need help understanding the concepts mentioned here.

To start, install `libcap2-bin`. Next, you need to grant QEMU the ability to administer networking by setting `cap_net_admin`. If you have a 64-bit system, use the following:

```
matthew@seymour:~$ sudo setcap cap_net_admin=ei /usr/bin/qemu-system-x86_64
```

If you have a 32-bit system, use this:

```
matthew@seymour:~$ sudo setcap cap_net_admin=ei /usr/bin/qemu
```

Then create a bridge interface called `enp3s0` under the bridges section in `/etc/netplan/*.yaml` by adding these lines to use DHCP or your network settings if you want to configure it yourself:

```
network:
  version: 2
  renderer: networkd
  ethernets:
    enp3s0:
      dhcp4: true
```

Restart networking by entering this:

```
matthew@seymour:~$ sudo netplan apply
```

Finally, you need to create guest VMs that use this bridged network. Manually define your guest OS to use the new `enp3s0` interface, as you usually would in that operating system.

There are several ways to create VMs for use with KVM. One way is to use `vmbuilder`. This is a Python script that is best for servers on which you intend to run a specialized, very light Ubuntu server variant that includes a tuned kernel with only the base elements necessary to run as a virtual server, especially under KVM and VMware. Install `python-vm-builder` to get the package. You run `vmbuilder` from the command line with two necessary parameters: the virtualization software and the distribution you will run. However, tons of useful options and customizations are available. Here is an example that builds a VM for KVM from the 18.04 (Bionic Beaver) release of Ubuntu, using the virtual flavor in an amd64 architecture while overwriting any previous edition of the VM, instructing `libvirt` to inform the local virtualization environment to add the resulting VM to the list of available virtual machines, give the new VM a specific IP address and the hostname `lovelace`, and use the `br0` bridge interface (Phew! That's a lot in one command!):

```
matthew@seymour:~$ sudo vmbuilder kvm ubuntu --suite bionic --flavour virtual --arch
amd64 -o --libvirt qemu:///system --ip 192.168.0.100 --hostname lovelace
--bridge br0
```

You can learn more from the help file:

```
matthew@seymour:~$ vmbuilder kvm ubuntu --help
```

Because `vmbuilder` is so specialized, here we focus on using the tools from `virtinst` as they are more likely to appeal to a general audience. However, if you are looking to create server VMs to run on a KVM or VMware installation, you definitely want to explore `vmbuilder` more fully. However, you might not need to do so. A set of official prebuilt and Ubuntu-supported VM images are available for download at <https://cloud-images.ubuntu.com>. These are the exact images that Ubuntu uses in EC2.

`virtinst` consists of several tools. Here we focus on two: `virt-install`, to provision new virtual machines; and `virt-clone`, to clone existing virtual machines.

You can do similar things with `virt-install` as with `vmbuilder`, shown earlier in this chapter. The major differences are the options available and that `virt-install` can also make desktop images that include a GUI, which is accessible using VNC. See Chapter 19, “Remote Access with SSH and VNC,” for a discussion of VNC.

Here is an example:

```
matthew@seymour:~$ sudo virt-install -n hopper -r 512 --disk
path=/var/lib/libvirt/images/hopper.img,size=20 -c /dev/cdrom --accelerate --
connect=qemu:///system --vnc --noautoconsole -v
```

This example includes the following options:

- ▶ **-n hopper**—Defines the name of the new VM
- ▶ **-r 512**—Specifies the amount of memory the virtual machine will be allotted, in megabytes
- ▶ **--disk path=...**—Specifies the path to the virtual disk, which can be a file, a partition, or a logical volume (in this case, a 20GB file named `hopper.img` in `/var/lib/libvirt/images`)
- ▶ **-c /dev/cdrom**—Specifies the path to the host's CD-ROM device (though you can also use an ISO file)
- ▶ **--accelerate**—Enables the use of the kernel's acceleration
- ▶ **--connect**—Defines the hypervisor to use
- ▶ **--vnc**—Exports the guest using a VNC virtual console
- ▶ **--noautoconsole**—Prevents automatic connection to the virtual machine's console
- ▶ **-v**—Creates a fully virtualized guest

To copy a virtual machine, use `virt-clone`:

```
matthew@seymour:~$ sudo virt-clone -o hopper -n knuth -f /var/lib/libvirt/images/knuth.img -connect=qemu:///system
```

This example includes the following options:

- ▶ **-o hopper**—Defines the name of the origin or source VM
- ▶ **-n knuth**—Defines the name of the new VM
- ▶ **-f**—Defines the path to the file, partition, or logical volume that the new VM will use
- ▶ **--connect**—Defines the hypervisor to use

To start a virtual machine, use the following:

```
matthew@seymour:~$ virsh -c qemu:///system start hopper
```

To stop a virtual machine, use this:

```
matthew@seymour:~$ virsh -c qemu:///system shutdown hopper
```

When a VM is installed and running, you can connect to it by using the configured IP address and a utility like `ssh`. You can also use a GUI with the following:

```
matthew@seymour:~$ virt-viewer -c qemu:///system hopper
```

You may use a GUI to manage your VMs by connecting to the following:

```
matthew@seymour:~$ virt-manager -c qemu:///system
```

VirtualBox

VirtualBox is much easier to use than KVM, especially if all you want to do is run a second operating system on top of Ubuntu. It was created by innotek GmbH, purchased by Sun Microsystems, and is now owned and developed by Oracle (which purchased Sun). VirtualBox is installed on top of another operating system, so it isn't ideal for processing intensive activity where every processor cycle counts. However, for testing or running another operating system because you need specific applications, it is great. VirtualBox runs on top of most UNIX-type operating systems, such as Linux, BSD, and macOS (as well as on Windows).

There is a version of VirtualBox in the Ubuntu software repositories, but in general downloading the one from the VirtualBox website is a better idea. Go to www.virtualbox.org/wiki/Downloads. From there, you can download a version for any operating system you are likely to use, including on the Linux page an Ubuntu DEB file that installs using the *Advanced Packaging Tool (APT)*, so package management isn't a problem. However, this also gives you easy access to download the extension pack, which isn't available in the Ubuntu repositories, so you can get it installed quickly and easily. The extension pack adds a few nice but proprietary features that cannot be made available under the GPL used for the main program, such as the ability to connect to the USB port of a host computer from a guest VM in VirtualBox. After you've installed it, start VirtualBox at the command line by using the following:

```
matthew@seymour:~$ virtualbox
```

If you want to be able to close your terminal and keep VirtualBox running, run VirtualBox in the background by putting an ampersand (&) after the command, like this:

```
matthew@seymour:~$ virtualbox &
```

Either way, when you are done, just close the GUI program, and VirtualBox shuts down. When you run either, the GUI appears (see Figure 30.1).

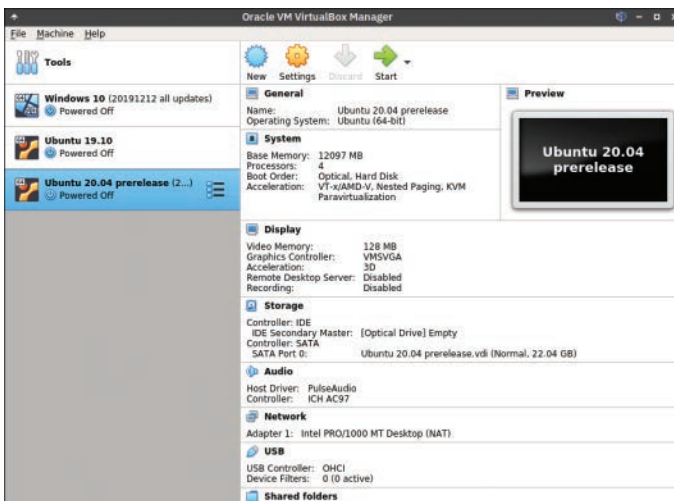


FIGURE 30.1 Oracle VM VirtualBox Manager.

From here you can create a new VM by clicking New at the top left. Change the settings on any currently installed VM by clicking Settings. Start any installed VM by clicking Start. Delete any VM by clicking Discard. You can see details of the currently selected VM by clicking Details at the upper right or your saved snapshots of existing VMs by clicking Snapshots. Everything is configurable from the GUI. VirtualBox is easy and intuitive to use, even for novices.

From the GUI, you can clone a machine, export it, and import it on another machine running VirtualBox. A command-line interface is available for scripting, focused on VM management activities. You can run VirtualBox headless and access it by using *Remote Display Protocol (RDP)*. These activities are beyond the scope of this book but can make VirtualBox a little more interesting to someone who wants to run VMs remotely (although most who are going to go through the trouble would probably go ahead and use KVM and connect to a VM using VNC).

VMware

VMware is an enterprise-focused virtualization platform. The company offers a limited-feature version that runs on the desktop for free and also sells a full-featured version. It runs well, is easy to use, and has better features than VirtualBox. It also requires buying a new license each year, and the license isn't cheap.

Many people consider VMware's enterprise server offerings to be the most powerful and well featured in the business. The VMware software runs on bare metal; it is the operating system that gets installed on all the servers in a VMware installation. Then all the resources are controlled from one central location. VMs can be moved while running from one physical machine to another in the system with no loss of usability and no downtime. This can even be done automatically, based on administrator-set criteria, such as bandwidth, available memory, or processor load. It is also quite expensive to license.

VMware is primarily designed for use by large corporations in enterprise environments. It is commonly used in large deployments by mature enterprise-level companies. Interestingly, VMware is expanding beyond virtualization with new offerings using newer cloud container services like Kubernetes.

Xen

Xen is a well-known open source virtualization platform. It is in widespread use by researchers, hobbyists, developers, and others. Web hosting companies that offer virtual servers often use Xen. Generally, Xen installs on bare metal, like VMware. It can be installed on top of another operating system in a host/guest arrangement. However, in 2008, Ubuntu made a decision not to support Xen. Instead, the Ubuntu community has focused its efforts on KVM. This is not a value statement that one is better than the other; it only means that KVM seemed to be a better fit for the needs of an Ubuntu developer community that did not have the resources to give quality support to two similar virtualization platforms.

It appears to be possible to run Xen on Ubuntu, but there are no guarantees. In fact, most Linux distributions have abandoned Xen for KVM now.

References

- ▶ www.linux-kvm.org—The main page for KVM
- ▶ www.virtualbox.org—The main page for VirtualBox
- ▶ www.vmware.com—The main page for VMware
- ▶ www.xenproject.org—The main page for Xen
- ▶ www.xenproject.org—The main page for Xen

This page intentionally left blank

CHAPTER 31

Containers and Ubuntu

The latest trend in cloud computing is containers. The easiest way to define containers is to begin with a comparison. Virtualization, which is described in Chapter 30, “Virtualization on Ubuntu,” enabled us to end our dependence on physical hardware when we need a new server; thanks to virtualization, we can instead create servers as virtual machines that can be moved across physical machines or parts of physical machines. Similarly, containers allow us to replicate the software components needed to run specific processes or programs. This is a much faster and smaller subset of software than virtualization requires.

A container packages an application and the application’s entire runtime environment—all the files and dependencies needed for the application to run. Containers can be large or small, depending on the use case and the software involved. Processes that run in a container are isolated from the rest of the system on which the container is running. This provides a level of security as well as extreme portability.

A trend in enterprise computing today that must not be ignored is the move to a microservice architecture. Microservice architecture attempts to modularize software system code into discrete chunks that can be easily replaced, updated, and replicated. The overall system is a collection of loosely coupled small (micro) services. The microservices are independently deployable and communicate with each other using a clearly defined mechanism, which may be private and internal to the system or publicly accessible (preferably via an API). Containers make it easier to create systems using this architecture.

Chapter 12, “Command-Line Master Class, Part 2,” has a section that describes a way to confine a script to a

IN THIS CHAPTER

- ▶ LXC and LXDM
- ▶ Docker
- ▶ Kubernetes
- ▶ References

directory, which is often called running in a *chroot jail*. This concept, borrowed from the FreeBSD world, can be used to prevent processes from accessing files or resources outside the directory (the *chroot jail*) in which the process is confined. You can think of containers as a stronger, more powerful, and better-isolated version of a *chroot jail*.

You need infrastructure, which could be a local machine such as a developer's laptop or a set of physical servers in a data center or a set of cloud-based servers running as virtual machines in one of the public clouds mentioned in Chapter 32, "Ubuntu and Cloud Computing."

On top of the infrastructure you run an operating system—in the case of this book and this chapter, we will assume Ubuntu. Over the past several years, changes to the Linux kernel (specifically the addition of control groups, or *cgroups*) and the development of a new initialization system called *systemd* (which uses *cgroups*) have expanded the ability to control and isolate user processes. Combining this with Linux kernel work on user namespaces, which allow the mapping of user and group IDs on a per-namespace basis, provides the foundation for containers. The user namespaces work allows a process to have root privileges within a defined user namespace while being a normal unprivileged process outside it. Because it is now possible to contain user and group privileges for operations to a certain subset of a system, the idea became known as running in a *container*.

The following sections of this chapter list and describe specific technologies. They are presented so that each section builds on the preceding one. There are "brands" and options out there besides the ones listed here, but this chapter presents prime examples of breakthrough technologies that have made the idea of containers progressively easier or more practical. As with some of the other overview-style chapters in this book, entire books could be and have been written about each of these, so we must content ourselves here with a high-level overview that provides enough information to give you a basic understanding and a few guideposts to help you discover where you might want to learn more.

LXC and LXD

Once user namespaces existed, it was possible to contain processes, but it still was not terribly easy or even practical to do this at any scale. The Linux Containers Project (<https://linuxcontainers.org>) created a set of tools, templates, and library and language bindings called *LXC* (sometimes pronounced "lexie") to make it easier to use the containment features of the Linux kernel and manage containers. The goal was to create an environment that comes as close as what one would have in a virtual machine but without the overhead of running a separate kernel and simulating all the hardware. The project is community based and completely open source, and it has been highly successful.

The next step was to create a command-line tool to make it easier to manage containers both locally and over a network. The Linux Containers Project created a next-generation container manager, which it called *LXD*. *LXD* builds on *LXC*, with an aim to improve the user experience.

In LXD, everything is image based, meaning an entire container is created, configured, and then stored. Then you can deploy an image or multiple copies of it, called *instances*, where you want to, knowing that all the instances are internally identical.

LXD containers are system containers, meaning each LXD container includes a full Linux system, exactly as it would if you were running it directly on a physical system or from within a virtual machine. This is a boon for cloud computing, especially when you want to automate the deployment of a large set of virtual servers across a data center or in a public cloud.

While LXD was founded by and development has been led by Canonical, it is open source, integrates with and runs on other Linux distributions, and has images available for deploying instances of CentOS, RHEL, SUSE, Debian, and others, along with Ubuntu instances.

We are not yet at the granular scale of “just the application and its runtime environment.” That comes next.

Docker

The Docker platform confusingly shares its name with a company created around a platform called Docker, Inc. The company supports and leads the development of the platform, which has open source projects that feed into the free Docker platform community edition and its for-payment enterprise Docker platform product. From here on, assume that we are referring to the Docker platform when we speak of *Docker*.

Docker is a container system with tools for building, deploying, and running application containers. The containers are everything the idea promised.

With Docker, you create a container image that includes only an application and the runtime software and dependencies needed to run that application. Once you have a container image created, you can deploy one or multiple instances of it to any machine or set of machines (physical or virtual) that are running the Docker Engine.

One of the greatest advantages of Docker containers is the portability of container instances. Each container instance is identical to the original image. The Docker Engine can be run locally on a developer’s machine, in an isolated development or testing environment, and in production environments. The application in the container instance should work identically in each location because details such as networking, storage, and operating system details are abstracted away.

Typically, a Docker container is created for each separate component in a larger application or system. This gives you the freedom to update or replace components without taking the entire system offline. In fact, you could deploy a new component instance while an older version of the instance is running, run them side-by-side for a while to compare and test, and then switch over with little to no downtime—at least in theory. You could deploy additional instances of a container instance when you discover that your current load is greater than your system can handle easily, notify your load balancer to add the additional instances to the pool of resources, and recover from an overload quickly. Much depends on your design and deployment details.

Kubernetes

As you can imagine, all the convenience of containers comes with a price: complexity. You must find a way to keep track of each containerized application, where it is deployed, and so on. To make containers useful without going insane, you will want to use a container platform that helps you develop, deploy, and manage applications (or services) across physical, virtual, and cloud-based servers. Enter Kubernetes.

Kubernetes is based on ideas rooted in years of experience running production workloads in containers at Google. Google has billions of containers running applications across data centers all over the world. Kubernetes is an open source system for automating deployment and management of containerized applications at whatever scale may be needed, up to Google-sized complexity. Kubernetes controls scaling, service discovery, load balancing, and a ton more. It is fully open source and built through cooperation between many large partners and a community of developers.

Kubernetes has quickly become the de facto standard for container management, and you can run Kubernetes on myriad different platforms. It is generally used in a cloud-type environment, whether that means within your private data center or a public cloud or a hybrid of the two.

At this point, the complexity involved in getting things set up and running is such that most companies hire experts to come in and help them get running.

References

- ▶ <https://linuxcontainers.org/lxd/>—The main website for LXD
- ▶ www.docker.com—The main website for Docker
- ▶ <https://kubernetes.io>—The main website for Kubernetes
- ▶ www.ubuntu.com/kubernetes—The main web page for Kubernetes via Canonical

CHAPTER 32

Ubuntu and Cloud Computing

Cloud computing enables you to build large, flexible systems for on-demand processing of data. When your requirements are low, you use few resources. As the need arises, your processes scale to use multiple systems with optimized performance according to the requirements of the moment. This is an efficient way to use hardware and minimize waste.

To accomplish this feat of computer engineering, a special network is set up using on-demand virtual systems that consume resources only as needed and release those resources for use by others when they are not in use. Virtualization is the technology that enables this concept. It may be accomplished locally using third-party virtualization platforms such as VMware, VirtualBox, Parallels, and others (see Chapter 30, “Virtualization on Ubuntu”). Ubuntu has another option to offer, the Ubuntu Cloud, which moves virtualization into the cloud and is the main focus of this chapter. Beyond being an outstanding cloud-hosting platform, Ubuntu Server is being developed with a strong intent to make it an outstanding cloud guest. Look for the term *Ubuntu cloud guest* to become more popular as time goes by.

SYSADMIN VERSUS DEVOPS

The traditional title for someone who keeps systems up and running is *systems administrator*, or *sysadmin* (sometimes called *ops*, for *operations*). The traditional title for someone who creates the software that runs on those systems is software developer. Over the past few years, a new title has emerged, *DevOps*. DevOps combine many of the talents and responsibilities of sysadmins and developers but often with a cloud computing environment focus and some specific refinements. They aren't purely one or

IN THIS CHAPTER

- ▶ Why a Cloud?
- ▶ Ubuntu on the Public Cloud
- ▶ Canonical-Specific Cloud Offerings
- ▶ References

the other and often don't fit neatly into other existing categories, such as engineer, but they do combine many of the skills of all of these while adding a quality assurance–like focus on making sure new features do not break anything that was working previously. DevOps are the ones who develop large applications to run on cloud resources while simplifying the orchestration of those resources with automation and configuration management. This chapter is not only for DevOps, but it describes the sorts of tools and environments that these folks are likely to love.

Ubuntu Cloud is a stack of applications from Canonical that are included in the Ubuntu Server Edition. These applications make it easy to install and configure an Ubuntu-based cloud. The software is free and open source, but Canonical offers paid technical support.

INSTALL INSTRUCTIONS

Install instructions change regularly, and the most up-to-date version is always on the provider's site. Instead of walking through the install instructions, this chapter provides a high-level view to help you understand how a cloud can be set up and work, why you should care, who the big players are, and where to look for the next steps.

Why a Cloud?

Businesses and enterprises have built computer networks for years. There are many reasons, but usually networks are built because specific computation or data processing tasks are made easier and faster by using more than one computer. The size of a network generally depends on the tasks that need to be done. Building a network usually entails taking a detailed survey of needs, analyzing those requirements, and gathering together the necessary hardware and software to fulfill those needs now, perhaps with a little room for growth if money permits.

Cloud computing is designed to make that easier by providing resources such as computing power and storage as services on the Internet in a way that is easy to access remotely, available on demand, simple to provision and scale, and highly dynamic. In the ideal case, this saves both time and money.

Some of the greatest benefits are the ease with which new resources may be added to a cloud, the fault tolerance inherent in the built-in redundancy of a large pool of servers, and the payment schedules that charge for resources only when they are used. There is also a great benefit in abstracting the complexity out of the process; clients perform the tasks they want to perform, and the cloud computing platform takes care of the details of adding resources as needed, without the end user's being aware of the process. *Virtual machines* (VMs) are created, configured, and used when needed and destroyed immediately after they are no longer needed, freeing up system resources for other purposes. These VMs can be created to suit a wide range of needs.

Hardware, storage, networks, and software are abstracted as services instead of being manually built and configured. They are then accessed locally on demand when the additional

resources are required. Sometimes these service model abstractions are referred to as *software as a service (SaaS)*, *platform as a service (PaaS)*, and *infrastructure as a service (IaaS)*.

Software as a Service (SaaS)

SaaS is sometimes referred to as *on-demand software*. In this service model, the software application and its related data are moved to the cloud. Access is generally through a web browser, although a thin client and server configuration may also be used. Someone else takes care of everything else. This is kind of like renting a hotel room: Everything is provided and set up for you, and you just enjoy and use it for a specific need. Some examples of SaaS include email hosts like Yahoo! Mail, services like Google Docs, web games, and customer relationship management (CRM) software.

Platform as a Service (PaaS)

PaaS takes things a step further than SaaS. In this service model, an entire computing platform is provided in the cloud. It's basically paying to run your containers on someone else's container platform. This typically includes the operating system, programming language interpreters or execution environments, databases, web servers, and so on. These facets are accessed directly for computing platform maintenance, using provider portals, application programming interfaces (APIs), software development kits (SDKs), or services like SSH. What is built on the platform is then accessed by the end user the same way it would be accessed if it were running on a locally owned and operated piece of hardware or hardware that's running in a large data center. Someone else takes care of everything else, but they take care of less than they do with SaaS, which means you take care of more. This scenario is rather like an apartment: You rent the space and decorate and configure it as you like, within structured guidelines. Some examples of PaaS include the Google App Engine, raw compute nodes used to scale services, and social application platforms like Facebook.

Infrastructure as a Service (IaaS)

IaaS goes even further than PaaS. In this service model, you transition your entire server to the cloud. Your provider offers computers, typically virtual ones, on which you can install any operating system (perhaps within a set menu the provider allows), and you can configure it as you like. Someone else takes care of the physical machines and networks, and you take care of all the rest. IaaS is like buying a condominium: You own it and can do whatever you want inside it, but someone else takes care of the grounds and landscaping.

Metal as a Service (Maas)

Generally, the only other step available beyond IaaS is traditional server building, where you are responsible for the physical machine and everything on it. However, Ubuntu has added another service to the list: *metal as a service (Maas)*, which is designed to bring the language of the cloud to physical servers. Their goal is to make it as easy to set up the physical hardware, deploy your app or service, and scale up or down dynamically as it is in the cloud. The server is installed on the physical hardware and then managed using one web interface to manage all the various machines. Learn more at <https://maas.io/>.

Things to Consider Before You Make Any Decisions

You don't have to create your own cloud infrastructure, but you can. You can also deploy Ubuntu to cloud providers such as Microsoft Azure or Amazon Web Services (AWS). Before you do anything like this, though, you need to carefully consider what your needs are and decide what sort of service(s) you need. Do you just want to run a web application on someone else's already-set-up server, or do you want to set up a system for scalable computing where additional Hadoop nodes can be added and removed at will when big jobs start and end? Only you know the answers to such questions. When you have it all figured out, you can seek your solution and can think about how you can use Ubuntu to set it up. This chapter describes many options, but you are the one who is in control. That is a powerful—and sometimes overwhelming—position. Thought and planning prevent painful mistakes and repeated engineering.

Ubuntu on the Public Cloud

There are several options to choose from when looking to move to cloud computing. Each has its unique methodologies, strengths, and command set. An in-depth study of each is beyond the scope of this book, but we are able to present a description of the big players providing public cloud access in North America and Europe (in Asia, look into Alibaba Cloud at <https://us.alibabacloud.com/>).

All of them are pay-to-play and use proprietary software, with the exception of OpenStack, but OpenStack is designed primarily as the foundation for a private cloud and requires a place for you to host it, such as your own data center or one of the providers they mention in the following link. That requires more local work and systems administration, which is what most companies moving to the cloud are trying to get away from. It can be incredibly useful in a hybrid cloud, which is an option described at the end of this section.

Learn more about Ubuntu and public cloud use at <https://ubuntu.com/public-cloud>.

OpenStack

OpenStack is an Apache-licensed cloud computing platform. It was founded as a collaboration between NASA and Rackspace. After less than a year, it boasted a worldwide community of developers. Adoption has been swift, and already many large corporations, universities, and institutions are using OpenStack for cloud computing. Ubuntu and OpenStack have worked closely together for a long time and have similar release schedules, and Ubuntu is the reference operating system for OpenStack.

OpenStack is not a service provider. It doesn't operate systems or data centers. OpenStack is open source software for building public and private and hybrid clouds. Many companies have implemented and use OpenStack, which is a good thing. It means that if you develop your cloud deployment and it works on one company's servers, if that company is using OpenStack, you can move that deployment to another company's servers with little or possibly no changes if the second company is also running OpenStack. In fact, it is easy enough to create your deployment across several different providers, using cloud servers from multiple companies concurrently, according to your needs.

For a current list of cloud providers offering OpenStack to customers for cloud deployments, see www.openstack.org/marketplace/public-clouds/. You will find big names along with many you have not yet heard of that may be just as suitable or perhaps even better for your needs.

Amazon Web Services

Amazon Web Services (AWS) is a mature, enterprise-quality cloud provider that has been around since the beginning. It hosts giants like Netflix and PayPal. It has tons of features, options, tools, and can be intimidating at first. It is the main cloud provider today with a number of accounts that far surpasses any competitor. It can handle any configuration you want or need and comes with great documentation and support. Ubuntu images are available in AWS for immediate deployment, making it quick and easy to get started. Learn more at <https://aws.amazon.com/>.

Google Cloud

Google Cloud is the second most popular cloud and also hosts some big clients like Target and the Home Depot. As with AWS, it is a mature offering with a rich toolset and myriad options available. Ubuntu images are also available in Google Cloud for immediate deployment. Learn more at <https://cloud.google.com/>.

Microsoft Azure

Microsoft entered the cloud services competition a bit later than the other two big names, but it has done so with gusto. Azure is a cloud provider boasting more deployment/hosting regions than any other provider. It includes a strong toolset and also has Ubuntu images available for immediate deployment. Learn more at <https://azure.microsoft.com/>.

Hybrid Clouds

There are two ways to deploy Ubuntu in the cloud: on a private cloud or on a public cloud. Both have benefits and drawbacks. This section presents the factors you need to consider when choosing. We also look at a way to mix the two, in what is called a *hybrid cloud*.

A *public cloud* is built on a cloud provider's systems. This means your local hardware requirements are minimal, your startup costs are low, deployment is quick, and growth is easy. A public cloud can be incredibly useful for testing and has gained the stability and reputation for also being a great idea for production. The drawback to working this way is that you do not physically control the hardware on which your cloud is running. For many this is a benefit, but such a situation might not be suitable for high security needs. Although you alone control the software and processes on your public cloud, there might be some worry about who has access to the machines. Although a cloud provider would not last long in business if its data centers and machines were not secure, some applications and data are so sensitive that you cannot afford to allow *any* outside risk. Legal constraints, such as from the Sarbanes-Oxley Act, sometimes force IT policy decisions in an organization and make the public option impossible.

A *private cloud* is created on hardware you own and control. This requires a large up-front commitment, but you have the security of running everything behind a company firewall and with complete knowledge of who is able to physically access your machines and who is listening on the network.

One thing to consider is the possibility of a hybrid cloud by starting as a private cloud and then creating interfaces from there to public services. Perhaps you prefer to keep some of your data and services stored on the private cloud, but you have other data that is less sensitive and want to use some services and applications on a public cloud. This is an avenue worth exploring if your company has a mixture of “must be secured and held in-house” and “we still want to keep it away from prying eyes, but if something happens, it won’t be catastrophic” needs. The big issue with this method is moving data between public and private servers; if you have large amounts of data that may move between the two, this can be prohibitive. As always, do your due diligence.

Canonical-Specific Cloud Offerings

Canonical offers some software that helps manage cloud deployments. They also sell service and consulting to help you use them.

Juju

Juju has been described as APT for the cloud. As you learned in Chapter 9, “Managing Software,” APT does an amazing job of installing, configuring, and starting complicated software stacks and services—but only as long as all of that happens on only one system. Juju extends this ability across multiple machines. Often, Linux servers are set up for similar tasks. Multiple physical machines may be deployed with similar configurations to work with one another in a network, perhaps for load distribution or redundancy to prevent downtime in the event of one failing or being overloaded. Systems administrators are masters at creating and orchestrating these networks. However, doing so traditionally requires setting up each machine individually, configuring its software settings, and so on.

Tools have appeared over the years to help with this great task, such as Chef and Puppet; see Chapter 33, “Managing Sets of Servers,” for a little more about these tools. Juju does for servers what package managers do for individual systems: It enables you to deploy services quickly and easily across multiple servers, simplifying the configuration process, and is particularly designed with cloud servers in mind. As with Chef’s recipes, those services are deployed using formulas that standardize communication, for example, and may have been written by different people.

What makes Juju different from Chef and Puppet is that the Juju formulas, called *charms*, encapsulate services, defining all the ways that services need to expose or consume configuration data to or from other services. This can be done many ways in the Juju charm, including via shell scripts or using Chef itself in solo mode. Also, Juju orchestrates provisioning by tracking its available resources (such as EC2, Eucalyptus, or OpenStack machines) and adding or removing them as appropriate.

Juju is pretty cool, but it hasn't seen much serious adoption outside Canonical, especially now that the OpenStack tools are growing in number and scope and with the rise of public clouds. However, it does have some unique features and is definitely worth your consideration. Check out <https://jujucharms.com> for more information.

Mojo: Continuous Delivery for Juju

Mojo, made by Canonical, helps you with configuration and tools to verify the success of Juju deployments. It gives you a structured means of having an entirely repeatable deployment process. More information is available at <https://mojo.canonical.com>.

Landscape

Landscape is an enterprise-focused systems management and monitoring tool that is available from Canonical. It can monitor Ubuntu Cloud servers like the ones discussed in this chapter. Landscape can be deployed locally on your cloud or used as part of a paid service from Canonical called *Ubuntu Advantage*. Landscape is described further in Chapter 33.

References

- ▶ www.ubuntu.com/cloud—The official Ubuntu introduction to cloud computing
- ▶ www.openstack.org—The official website for OpenStack
- ▶ <https://landscape.canonical.com>—Canonical's Landscape, a commercial management tool for Ubuntu Cloud and Amazon EC2 instances
- ▶ <https://jujucharms.com>—The official Ubuntu documentation for Juju
- ▶ <https://conjure-up.io>—The official site for Conjure Up, which is used to deploy big software stacks to the cloud using Juju
- ▶ <https://maas.io>—The official documentation for Ubuntu and MAAS

This page intentionally left blank

CHAPTER 33

Managing Sets of Servers

This chapter provides a quick introduction to some tools that might be useful, especially for people who manage large numbers of servers. The tools presented here are designed for system administrators and developers—people who are responsible for keeping more than one or two servers up and running, managing their configurations, and so on. Systems have grown to the point that managing each one individually is becoming unwieldy at times, and so tools have been created to decrease some of the stress and complexity by reducing the job to managing the software that in turn manages all the servers. Any system administrator or developer will tell you that this is a bit of an oversimplification, but it is close.

Because managing sets of servers is such a complex task—one on which many thick books are written—we just provide a quick overview of some of the options you should consider and then leave you the task of further research. Some of these options have entire books just to themselves, but if you are someone who needs to manage sets of servers, you will want to do it right. Do your research. This chapter helps you get started with a quick description of some of the tools and technologies that are being used with Ubuntu to assist with managing servers, listed in order of popularity.

Puppet

Puppet is a configuration management tool written in Ruby. It is designed to make it easier to deploy servers and scale applications across a network, and it does so using a custom declarative language. It has both open source and commercial (closed source, or proprietary) versions.

IN THIS CHAPTER

- ▶ Puppet
- ▶ Chef
- ▶ Ansible
- ▶ SaltStack
- ▶ CFEngine
- ▶ Juju
- ▶ Landscape
- ▶ References

Puppet does four basic things. First, it provides a place for you to define the desired state for your infrastructure's configuration. Second, Puppet simulates changes before enforcing them. Third, Puppet enforces the desired state automatically, making corrections for drifting configurations. Finally, Puppet gives you a report on the differences between the actual and desired states before you make any changes.

The Puppet Forge website (<https://forge.puppetlabs.com>) provides access to downloadable *modules*, which are bits of Puppet code for automating tasks, such as setting up a specific type of server. Instructions there are also available for creating and sharing your own modules.

Much of the power of Puppet is made available in its for-payment Enterprise version, which also includes a nice GUI. The open source version works with Amazon for provisioning, manages configurations for operating systems and applications, lets you use Puppet modules from Puppet Forge, and has community support via the Web. If you need anything more than this, either find a different product or pay to use Puppet's Enterprise version, which has a good reputation for maturity and usefulness.

Chef

Chef is a configuration management tool, also written in Ruby. It uses a service-oriented architecture to help automate tasks. With Chef, you write recipes that describe how you want your server or specific server software to be configured (for example, an HTTP server or a database server). A recipe describes each resource, such as services that should be running or packages that should be installed, and the state in which each should be found. It then makes sure that configuration is maintained or updated across all servers being managed.

Chef is open source. However, to really use it, you must pay for a hosted version (called Hosted) or a standalone version (called Private) that you can install inside your corporate firewall with a service contract. Downloading the source code and getting it up and running is not a trivial task, and support for the open-source version (called Open Source) without a service contract is limited. That said, like Puppet, Chef is mature and well respected and worth your time if you need what it offers.

Ansible

Ansible is an orchestration engine. It does configuration management, application deployment, and more. It was a proprietary product, but Red Hat bought the company that created it and promptly worked its magic to release the code with an open source license. Much the way Red Hat handles its enterprise Linux and middleware products, it provides free and open source Ansible code—but only in source form. You can license binaries from Red Hat, which the company will then support. Ansible is well respected and worth looking into.

SaltStack

SaltStack automates infrastructure, networking, and security. It works both on premises and in the cloud and at scale. It is popular (second only to Ansible at the time of this

writing) and does things that are tedious and important, including continuous compliance policy checks and native CVE scanning, including automated remediation in many instances.

CFEngine

CFEngine is probably the oldest option for automating infrastructure. It is written in C, so it might be a little faster than the other options. It manages server builds, deployment, and management, and it performs some very useful audits and reports. Some really big names are known to use CFEngine, such as AT&T, IBM, Pixar, and Qualcomm.

CFEngine has open source and enterprise versions. The differences are a little less extreme than with some of the other options, but again, the enterprise version has all the flashy chrome and a few really useful features that make it worth the expenditure for most users who need or want the added benefits they offer.

Juju

Juju enables you to deploy services quickly and easily across multiple servers, simplifying the configuration process, and is particularly designed with cloud servers in mind. It is described in Chapter 32, “Ubuntu and Cloud Computing.”

Landscape

Landscape is an enterprise-focused systems management and monitoring tool that is available from Canonical. It is a part of the Ubuntu Advantage program (www.canonical.com/services), which is a paid service. You can run Landscape as a hosted service from Canonical’s Ubuntu Advantage, or you can install it locally. It can monitor both local servers and cloud servers, such as those discussed in Chapter 32.

References

- ▶ www.puppetlabs.com—The official website for Puppet
- ▶ <https://forge.puppetlabs.com>—The official website for Puppet Forge
- ▶ www.chef.io/chef/—The official website for Chef
- ▶ www.saltstack.com/—The official website for SaltStack
- ▶ <https://cfengine.com>—The official website for CFEngine
- ▶ www.ansible.com—The official website for Ansible
- ▶ www.ubuntu.com/support—The official website for Ubuntu Advantage
- ▶ <https://help.ubuntu.com/20.04/serverguide/index.html>—The official Ubuntu server guide

This page intentionally left blank

CHAPTER 34

Handling Email

Email is still the dominant form of communication over the Internet. It is fast, free, and easy to use. However, much of what goes on behind the scenes is extremely complicated and would appear scary to anyone who does not know much about how email is handled. Ubuntu comes equipped with a number of powerful applications that will help you build anything from a small email server, right through to large servers capable of handling thousands of messages.

This chapter shows you how to configure Ubuntu to act as an email server. We look at the options available in Ubuntu and examine the pros and cons of each one. You also learn how mail is handled in Linux and, to a lesser extent, in UNIX.

How Email Is Sent and Received

Email is transmitted as plain text across networks around the world using the *Simple Mail Transfer Protocol (SMTP)*. As the name implies, the protocol itself is fairly basic, and it has been extended to add further authentication and error reporting/messaging to satisfy the growing demands of modern email. *Mail transfer agents (MTAs)* work in the background, transferring email from server to server, allowing emails to be sent all over the world. You might have come across MTA software such as Sendmail, Postfix, Fetchmail, Exim, or Qmail.

SMTP allows each computer that the email passes through to forward it in the right direction toward the final destination. When you consider the millions of email servers around the world, you have to marvel at how simple it all seems.

IN THIS CHAPTER

- ▶ How Email Is Sent and Received
- ▶ Basic Postfix Configuration and Operation
- ▶ Using Fetchmail to Retrieve Mail
- ▶ Choosing a Mail Delivery Agent
- ▶ References

Here is a simplified example of how email is successfully processed and sent to its destination:

1. `matthew@seymourcray.net` composes and sends an email message to `sandra@gracehopper.net`.
2. The MTA at `seymourcray.net` receives Matthew's email message and queues it for delivery behind any other messages that are also waiting to go out.
3. The MTA at `seymourcray.net` contacts the MTA at `gracehopper.net` on port 24. After `gracehopper.net` acknowledges the connection, the MTA at `seymourcray.net` sends the mail message. After `gracehopper.net` accepts and acknowledges receipt of the message, the connection is closed.
4. The MTA at `gracehopper.net` places the mail message into Sandra's incoming mailbox; Sandra is notified that she has new mail the next time she logs on.

However, several things can go wrong during this process. Here are a few examples:

- ▶ What if Sandra does not exist at `gracehopper.net`? In this case, the MTA at `gracehopper.net` rejects the email and notifies the MTA at `seymourcray.net` about the problem. The MTA at `seymourcray.net` then generates an email message and sends it to `matthew@seymourcray.net`, informing him that no Sandra exists at `gracehopper.net` (or perhaps just silently discards the message and gives the sender no indication of the problem, depending on how the email server is configured).
- ▶ What happens if `gracehopper.net` doesn't respond to `seymourcray.net`'s connection attempts? (Perhaps the server is down for maintenance.) The MTA at `seymourcray.net` notifies the sender that the initial delivery attempt has failed. Further attempts will be made at intervals decided by the server administrator until the deadline is reached, and the sender will be notified that the mail is undeliverable.

The Mail Transport Agent

Several MTAs are available for Ubuntu, each with its pros and cons. Normally they are hidden under the skin of Ubuntu, silently moving mail between servers all over the world, with little or no maintenance required. Some MTAs are extremely powerful, able to cope with hundreds of thousands of messages each day, whereas some are geared more toward smaller installations. Other MTAs are perhaps not as powerful but are packed full of features. In this section, we take a look at some of the most popular MTAs available for Ubuntu.

Sendmail

Sendmail handles the overwhelming majority of emails transmitted over the Internet today. It is extremely popular across the Linux/UNIX/BSD world and is well supported. A commercial version is available that has a GUI interface for ease of configuration.

As well as being popular, Sendmail is particularly powerful compared to some of the other MTAs. However, it is not without downsides, and you will find that other MTAs can handle more email per second in a larger environment. The other issue with Sendmail is that it can be extremely complicated to set it up exactly as you want it. A few books are available specifically for Sendmail; the most popular one has more than 1,000 pages, which gives you an idea of the complex nature of the Sendmail configuration.

We can be thankful, however, that the default configuration for Sendmail works fine for most basic installations out of the box, making further configurations unnecessary. Even if you want to use it as a basic email server, you only need to do some minor tweaks. The level of complexity associated with Sendmail often leads to system administrators replacing it with one of the other alternatives that is easier to configure.

Postfix

Postfix has its origins as the IBM Secure Mailer but was released to the community by IBM. Compared to Sendmail, it is much easier to administer and has a number of speed advantages. Postfix offers a pain-free replacement for Sendmail, and you can replace Sendmail with Postfix without the system breaking a sweat. In fact, the applications that rely on Sendmail automatically use Postfix instead and carry on working correctly (because Postfix uses a Sendmail wrapper, which deceives other programs into thinking that Postfix is Sendmail). This wrapper—or, more correctly, interface—makes switching to Postfix extremely easy if you are already running Sendmail. Postfix also happens to be the MTA of choice for Ubuntu, so it is this one that we spend more time on later in this chapter.

For enhanced security, many Postfix processes used to use the `chroot` facility (which restricts access to only specific parts of the file system) for improved security, and there are no `setuid` components in Postfix. In Ubuntu, a `chroot` configuration is *no longer used* and is, in fact, discouraged by the Postfix author. You can manually reconfigure Postfix to a `chroot` configuration, but that is no longer supported by Ubuntu.

If you are starting from scratch, Postfix is considered a better choice than Sendmail.

Qmail and Exim

Qmail is a direct competitor to Postfix but is not provided with Ubuntu. Qmail is designed to be easier to use than Sendmail, as well as faster and more secure. However, Qmail is not a drop-in replacement for Sendmail, and migrating an existing Sendmail installation to Qmail is not quite as simple as migrating from Sendmail to Postfix. Qmail is relatively easy to administer, and it integrates with a number of software add-ons, including web mail systems and POP3 servers. Qmail is available from www.qmail.org.

Exim is yet another MTA, and it is available at www.exim.org. Exim is considered faster and more secure than Sendmail or Postfix but is quite different to configure than either of those. Exim and Qmail use the `maildir` format rather than `mbox`, so both are considered “NFS safe” (see the following sidebar).

Maildir Versus Mbox

Qmail introduced `maildir`, which is an alternative to the standard UNIX method of storing incoming mail. `maildir` is a more versatile system of handling incoming email, but it requires your email clients to be reconfigured, and it is not compatible with the traditional UNIX way of storing incoming mail. You need to use mail programs that recognize the `maildir` format (which modern programs do).

The traditional `mbox` format keeps all mail assigned to a folder concatenated as a single file and maintains an index of individual emails. With `maildir`, each mail folder has three subfolders: `/cur`, `/new`, and `/tmp`. Each email is kept in a separate, unique file. If you are running a mail server for a large number of people, you should select a file system that can efficiently handle a large number of small files.

`mbox` does have one major disadvantage. While you are accessing the monolithic `mbox` file that contains all your email, suppose that some type of corruption occurs, either to the file itself or to the index. Recovery from this problem can be difficult. The `mbox` files are especially prone to problems if the files are being accessed over a network and can result in file corruption; you should avoid accessing `mbox` mail mounted over *Network File System (NFS)* because file corruption can occur.

Depending on how you access your mail, `maildir` does permit the simultaneous access of `maildir` files by multiple applications; `mbox` does not.

The choice of a *mail user agent (MUA)*, or email client, also affects your choice of mail directory format. For example, the `pine` program does not cache any directory information and must reread the mail directory any time it accesses it. If you are using `pine`, `maildir` is a poor choice. More advanced email clients perform caching, so `maildir` might be a good choice, although the email client cache can get out of synchronization. It seems that no perfect choice exists.

Ubuntu provides mail alternatives that have both strong and weak points. Be aware of the differences among the alternatives and frequently reevaluate your selection to make certain it is the best one for your circumstances.

Choosing an MTA

Other MTAs are available for use with Ubuntu, but those discussed in the previous sections are the most popular. Which one should you choose? It depends on what you need to do. Postfix's main strengths are that it scales well and can handle large volumes of email at high speeds, not to mention that it is much easier to configure than the more cryptic Sendmail. However, you may find that there are specific things you need that only Sendmail can provide. It is easy to switch between MTAs when you need to.

The Mail Delivery Agent

SMTP is a server-to-server protocol that was designed to deliver mail to systems that are always connected to the Internet. Dial-up systems connect only at the user's command; they connect for specific operations and are frequently disconnected. To accommodate this difference, many mail systems also include a *mail delivery agent (MDA)*. The MDA transfers mail to systems without permanent Internet connections. The MDA is similar to an MTA (see the following note) but does not handle deliveries between systems and does not provide an interface to the user.

NOTE

Procmail or Spamassassin are examples of MDAs; both provide filtering services to the MDA while they store messages locally and then make them available to the MUA or email client for reading by the user.

The MDA uses the *Post Office Protocol version 3 (POP3)* or *Internet Message Access Protocol (IMAP)* for this process. In a manner similar to a post office box at the post office, POP3 and IMAP implement a “store and forward” process that alleviates the need to maintain a local mail server if all you want to do is read your mail. For example, dial-up Internet users can intermittently connect to their ISP’s mail server to retrieve mail using Fetchmail—the MDA recommended by Ubuntu. (See the section “Using Fetchmail to Retrieve Mail,” later in this chapter.)

The Mail User Agent

The *mail user agent (MUA)* is another necessary part of the email system. The MUA is a mail client, or mail reader, that enables the user to read and compose email and provides the user interface. (It is the email application itself that most users are familiar with as “email.”) Some popular UNIX command-line MUAs are `elm`, `pine`, and `mutt`. Ubuntu also provides modern GUI MUAs: Evolution, Thunderbird, Mozilla Mail, Balsa, Sylpheed, and KMail. For comparison, common non-UNIX MUAs are Microsoft Outlook, Outlook Express, Pegasus Mail, and Apple Inc.’s Mail.

The Microsoft Windows and Macintosh MUAs often include some MTA functionality; UNIX does not. For example, Microsoft Outlook can connect to your Internet provider’s mail server to send messages. On the other hand, UNIX MUAs generally rely on an external MTA such as Sendmail. This might seem like a needlessly complicated way to do things, and it is, if used to connect a single user to her ISP. For any other situation, however, using an external MTA allows you much greater flexibility because you can use any number of external programs to handle and process your email functions and customize the service. Having the process handled by different applications gives you great control over how you provide email service to users on your network, as well as to individual and *small office/home office (SOHO)* users.

For example, you could do the following:

- ▶ Use Evolution to read and compose mail.
- ▶ Use Sendmail to send your mail.
- ▶ Use `xbiff` to notify you when you have new mail.
- ▶ Use Fetchmail to retrieve your mail from a remote mail server.
- ▶ Use Procmail to automatically sort your incoming mail based on sender, subject, or many other variables.
- ▶ Use Spamassassin to eliminate the unwanted messages before you read them.

Basic Postfix Configuration and Operation

Because Postfix is the Ubuntu-recommended MTA, the following sections provide a brief explanation and examples for configuring and operating your email system. As mentioned earlier, however, Postfix is an extremely complex program with many configuration options. Therefore, this chapter covers only some of the basics.

Postfix is not installed by default. To use it, install the `postfix` package from the Ubuntu software repositories. During installation, you are asked a series of questions to begin configuring Postfix immediately. Research the settings for your situation before you start; some are merely preferential, but others are based on your hardware, network, and use case. You are asked the following:

```
General type of mail configuration: Internet Site

System mail name: mail.matthewhelmke.com
Root and postmaster mail recipient: <admin_user_name>
Other destinations for mail: mail.example.com, example.com, localhost.example.com,
[ccc]localhost
Force synchronous updates on mail queue?: No
Local networks: 127.0.0.0/8

Mailbox size limit (bytes): 0
Local address extension character: +
Internet protocols to use: all
```

If you make a mistake and answer a configuration question incorrectly, you can go back through the process again by entering the following command from the command line:

```
matthew@seymour:~$ sudo dpkg-reconfigure postfix
```

Postfix configuration is maintained in files in the `/etc/postfix` directory, and much of the configuration is handled by the file `main.cf`. You don't have to use the preceding command to change these settings; you may do so by editing the appropriate files. The syntax of the configuration file, `main.cf`, is fairly easy to read, as shown in the following example:

```
# See /usr/share/postfix/main.cf.dist for a commented, more complete version
# Debian specific: Specifying a file name will cause the first
# line of that file to be used as the name. The Debian default
# is /etc/mailname.
#myorigin = /etc/mailname

smtpd_banner = $myhostname ESMTPE $mail_name (Ubuntu)
biff = no

# appending .domain is the MUA's job.
append_dot_mydomain = no
```

```
# Uncomment the next line to generate "delayed mail" warnings
#delay_warning_time = 4h

# TLS parameters
smtpd_tls_cert_file=/etc/ssl/certs/ssl-cert-snakeoil.pem
smtpd_tls_key_file=/etc/ssl/private/ssl-cert-snakeoil.key
smtpd_use_tls=yes
smtpd_tls_session_cache_database = btree:${queue_directory}/smtpd_scache
smtp_tls_session_cache_database = btree:${queue_directory}/smtp_scache

# See /usr/share/doc/postfix/TLS_README.gz in the postfix-doc package for
# information on enabling SSL in the smtp client.

myhostname = optimus
alias_maps = hash:/etc/aliases
alias_database = hash:/etc/aliases
mydestination = optimus, localhost.localdomain, , localhost
relayhost =
mynetworks = 127.0.0.0/8
mailbox_size_limit = 0
recipient_delimiter = +
inet_interfaces = all
```

A useful command for configuring Postfix is `postconf`. It enables you to display and change many configuration settings without editing and saving configuration files. The command's syntax is rather complex, but once learned, it becomes a faster way to quickly adjust settings.

If you type the `postconf` command by itself, it outputs a list of all configuration parameters. This can be quite long, so we recommend either sorting using a pipe and `grep` or sending the output to a file. See Chapter 11, "Command-Line Master Class, Part 1," for more on how to do this. This example shows the command piped into `grep` with a search for `hostname`:

```
matthew@seymour:~$ postconf | grep hostname
invalid_hostname_reject_code = 501
lmtp_lhlo_name = $myhostname
lmtp_tls_verify_cert_match = hostname
local_transport = local:$myhostname
milter_macro_daemon_name = $myhostname
myhostname = ubuntu
smtp_helo_name = $myhostname
smtp_tls_verify_cert_match = hostname
smtpd_banner = $myhostname ESMTP $mail_name (Ubuntu)
smtpd_proxy_ehlo = $myhostname
unknown_helo_hostname_tempfail_action = $reject_tempfail_action
unknown_hostname_reject_code = 450
```


To show the default parameter settings instead of the current settings, use this:

```
matthew@seymour:~$ postconf -d
```

Use this to discover which parameters have been changed from their defaults and display the current settings:

```
matthew@seymour:~$ postconf -n
```

Setting a parameter requires root privileges. For example, to set the `myhostname` parameter, use the following:

```
matthew@seymour:~$ sudo postconf -e "myhostname=mail.matthewhelmke.com"
myhostname=mail.matthewhelmke.com
```

This works with the parameters listed in the Postfix `main.cf` file:

```
matthew@seymour:~$ sudo postconf -e "smtp_sasl_auth_enable = yes"
myhostname=othername.matthewhelmke.com
```

As you can see, `postconf` is quite convenient. You can learn more about `postconf` from the man page.

You start, stop, and restart Postfix by using this command, with the appropriate one of the three action words `start`, `stop`, or `restart`, as shown here:

```
matthew@seymour:~$ sudo /etc/init.d/postfix start
```

Complicated email server setup is beyond the scope of this book; consider *Postfix: The Definitive Guide* by Kyle Dent for more information. This is a great reference and rather unusual because it is a complete and useful reference in only 250 pages or so. If you want to know something specific about Postfix, this is the book to read.

The following sections address some commonly used advanced options. For more information on Postfix, as well as other MTAs, see the “References” section at the end of this chapter.

Configuring Masquerading

Sometimes you might want to have Postfix masquerade as a host with a name other than the actual hostname of your system. Such a situation could occur if you have a dial-up connection to the Internet and your ISP handles all your mail for you. In this case, you want Postfix to masquerade as the domain name of your ISP. For example, the following strips any messages that come from `matthew.gracehopper.net` to just `gracehopper.net`:

```
masquerade_domains = gracehopper.net
```

Using Smart Hosts

If you do not have a full-time connection to the Internet, you will probably want to have Postfix send your messages to your ISP’s mail server and let it handle delivery for you.

Without a full-time Internet connection, you could find it difficult to deliver messages to some locations (such as some underdeveloped areas of the world, where email services are unreliable and sporadic). In those situations, you can configure Postfix to function as a smart host by passing email on to another sender instead of attempting to deliver the email directly. You can use a line such as the following in the `main.cf` file to enable a smart host:

```
relayhost = mail.isp.net
```

This line causes Postfix to pass any mail it receives to the server `mail.isp.net` rather than attempt to deliver it directly. Smart hosting will not work for you if your ISP blocks any mail relaying. Some ISPs block relaying because it is frequently used to disseminate spam.

Setting Message Delivery Intervals

As mentioned earlier in this chapter, Postfix typically attempts to deliver messages as soon as it receives them, and it attempts to deliver them again at regular intervals after that. If you have only periodic connections to the Internet, as with a dial-up connection, you likely would prefer Sendmail to hold all messages in the queue and attempt to deliver them whenever you connect to your ISP.

As dialup connections have become the exception rather than the rule and are now quite rare, Ubuntu does not configure them by default and does not include the `pppd` daemon in the default installation. If you need this, install `pppd` from the Ubuntu software repositories. You can then configure Postfix to hold messages for later delivery by adding the following line to `/etc/ppp/peers/ppp0`:

```
/usr/sbin/sendmail -q
```

This line causes Postfix to automatically send all mail when you connect to your ISP.

However, Postfix still attempts to send mail regardless of whether the computer is online or offline, which means your computer may dial out just to send email. To disable this, you need to enter the following line into `mail.cf`:

```
defer_transports = smtp
```

This stops any unwanted telephone calls from being placed!

TIP

If you use networking over a modem, there is a configuration file for `pppd` called `ppp0`, which is located in `/etc/ppp/peers`. Any commands in this file automatically run each time the PPP daemon is started. You can add the line `sendmail -q` to this file to have your mail queue automatically processed each time you dial up your Internet connection.

Mail Relaying

By default, Postfix does not relay mail that did not originate from the local domain. This means that if a Postfix installation running at `gracehopper.net` receives mail intended for

seymourcray.net, and that mail did not originate from gracehopper.net, the mail is be rejected and is not relayed. If you want to allow selected domains to relay through you, add an entry for the domain to the `main.cf` file like this:

```
mynetworks = 192.168.2.0/24, 10.0.0.2/24, 127.0.0.0/8
```

The IP address needs to be specified in *classless interdomain routing (CIDR)* format. For a handy calculator, head on over to www.subnet-calculator.com/cidr.php. You must restart Postfix for this change to take effect.

CAUTION

You need a good reason to relay mail; if you don't have one, do not do it. Allowing all domains to relay through you will make you a magnet for spammers who will use your mail server to send spam. This can lead to your site being blacklisted by many other sites, which then will not accept any mail from you or your site's users—even if the mail is legitimate!

Forwarding Email with Aliases

Aliases allow you to have an infinite number of valid recipient addresses on your system, and you don't have to worry about creating accounts or other support files for each address. For example, most systems have `postmaster` defined as a valid recipient but do not have an actual login account named `postmaster`.

Aliases are configured in the file `/etc/aliases`. Here is an example of an alias entry:

```
postmaster: root
```

This entry forwards any mail received for `postmaster` to the `root` user. By default, almost all the aliases listed in the `/etc/aliases` file forward to `root`.

CAUTION

Reading email as `root` is a security hazard; a malicious email message can exploit an email client and cause it to execute arbitrary code as the user running the client. To avoid this danger, you can forward all of `root`'s mail to another account and read it from there. You can choose one of two ways to do this.

One way is to add an entry to the `/etc/aliases` file that sends `root`'s mail to a different account. For example, `root: foobar` would forward all mail intended for `root` to the account `foobar`.

The other way is to create a file named `.forward` in `root`'s home directory that contains the address that the mail should forward to.

Any time you make a change to the `/etc/aliases` file, you must rebuild the aliases database before that change will take effect. Use the following command to do this:

```
matthew@seymour:~$ sudo newaliases
```

Using Fetchmail to Retrieve Mail

SMTP is designed to work with a system that has a full-time connection to the Internet. What if you are on a dial-up account? What if you have another system store your email for you and then you log in to pick it up once in a while? (Most users who are not setting up servers are in this situation.) In this case, you cannot easily receive email using SMTP, and you need to use a protocol such as POP3 or IMAP instead.

NOTE

Remember when we said that some mail clients can include some MTA functionality? You can configure Microsoft Outlook and Outlook Express to use SMTP, and if you use a dial-up connection, they offer to start the connection and then use SMTP to send your mail, so a type of MTA functionality is included in those mail clients.

Unfortunately, many MUAs do not know anything about POP3 or IMAP. To eliminate that problem, you can use a program called Fetchmail to contact mail servers using POP3 or IMAP, download mail from the servers, and then inject those messages into the local MTA just as if they had come from a standard SMTP server. The following sections explain how to install, configure, and use the Fetchmail program.

Installing Fetchmail

Much as with other packages, you can install `fetchmail` using either `synaptic` or `apt`.

You can get the latest version of Fetchmail at www.catb.org/~esr/fetchmail.

Configuring Fetchmail

After you have installed Fetchmail, you must create the file `.fetchmailrc` in your home directory, which provides the configuration for the Fetchmail program.

You can create and subsequently edit the `.fetchmailrc` file by using any text editor. The configuration file is straightforward and quite easy to create; the following sections explain the manual method for creating and editing the file. The information presented in the following sections does not discuss all the options available in the `.fetchmailrc` file but covers the most common ones needed to get a basic Fetchmail installation up and running. You must use a text editor to create the file to include entries like the ones shown as examples—modified for your personal information, of course. For advanced configuration, see the man page for Fetchmail. The man page is well written and documents all the configuration options in detail.

CAUTION

The `.fetchmailrc` file is divided into three sections: global options, mail server options, and user options. It is important that these sections appear in the order listed. Do not add options to the wrong section. Putting options in the wrong place is one of the most common problems that new users make with Fetchmail configuration files.

Configuring Global Options

The first section of `.fetchmailrc` contains the global options. These options affect all the mail servers and user accounts that you list later in the configuration file. You can override some of these global options with local configuration options, as you learn later in this section. Here is an example of the options that might appear in the global section of the `.fetchmailrc` file:

```
set daemon 600
set postmaster foobar
set logfile ./fetchmail.log
```

The first line in this example tells Fetchmail that it should start in daemon mode and check the mail servers for new mail every 600 seconds, or 10 minutes. Daemon mode means that after Fetchmail starts, it moves itself into the background and continues running. Without this line, Fetchmail checks for mail once when it started and then terminates and never checks again.

The second option tells Fetchmail to use the local account `foobar` as a last-resort address. In other words, any email that it receives and cannot deliver to a specified account should be sent to `foobar`.

The third line tells Fetchmail to log its activity to the file `./fetchmail.log`. Alternatively, you can use the line `set syslog`—in which case Fetchmail logs through the syslog facility.

Configuring Mail Server Options

The second section of the `.fetchmailrc` file contains information on each of the mail servers that should be checked for new mail. Here is an example of what the mail section might look like:

```
poll mail.samplenet.org
proto pop3
no dns
```

The first line tells Fetchmail that it should check the mail server `mail.samplenet.org` at each poll interval that was set in the global options section (which was 600 seconds in this example). Alternatively, the first line can begin with `skip`. If a mail server line begins with `skip`, it will not be polled at the poll interval but will be polled only when it is specifically specified on the Fetchmail command line.

The second line specifies the protocol that should be used when contacting the mail server. This example uses POP3. Other legal options are IMAP, *Authenticated Post Office Protocol (APOP)*, and *Kerberized Post Office Protocol (KPOP)*. You can also use `AUTO` here, in which case Fetchmail attempts to automatically determine the correct protocol to use with the mail server.

The third line tells Fetchmail that it should not attempt to do a *Domain Name System (DNS)* lookup. You probably want to include this option if you are running over a dial-up connection.

Configuring User Accounts

The third and final section of `.fetchmailrc` contains information about the user account on the server specified in the previous section. Here is an example:

```
user foobar
pass secretword
fetchall
no flush
```

The first line, of course, simply specifies the username that is used to log in to the email server, and the second line specifies the password for that user. Many security-conscious people cringe at the thought of putting clear-text passwords in a configuration file, and they should if it is group or world readable. The only protection for this information is to make certain that the file is readable only by the owner—that is, with file permissions of `600`.

The third line tells Fetchmail that it should fetch all messages from the server, even if they have already been read.

The fourth line tells Fetchmail that it should delete the messages from the mail server after it has finished downloading them. This is the default, so you would not really have to specify this option. If you want to delete the messages from the server after downloading them, use the option `flush`.

The configuration options you just inserted configured the entire `.fetchmailrc` file to look like this:

```
set daemon 600
set postmaster foobar
set logfile ../fetchmail.log

poll mail.samplenet.org
proto pop3
no dns

user foobar
pass secretword
fetchall
flush
```

This file tells Fetchmail to do the following:

- ▶ Check the POP3 server `mail.samplenet.org` for new mail every 600 seconds.
- ▶ Log in using the username `foobar` and the password `secretword`.
- ▶ Download all messages from the server.
- ▶ Delete the messages from the server after downloading them.
- ▶ Send any mail it receives that cannot be delivered to a local user to the account `foobar`.

As mentioned earlier, many more options can be included in the `.fetchmailrc` file than are listed here. However, the options offered in this section will get you up and running with a basic configuration.

For additional flexibility, you can define multiple `.fetchmailrc` files to retrieve mail from different remote mail servers while using the same Linux user account. For example, you can define settings for your most commonly used account and save them in the default `.fetchmailrc` file. Mail can then quickly be retrieved like this:

```
matthew@seymour:~$ fetchmail -a
1 message for matthew at mail.matthewhelmke.com (1108 octets) .
reading message 1 of 1 (1108 octets) . flushed
```

By using Fetchmail's `-f` option, you can specify an alternative resource file and then easily retrieve mail from another server, as follows:

```
matthew@seymour:~$ fetchmail -f .myothermailrc
2 messages for matthew at matthew.helmke.com (5407 octets) .
reading message 1 of 2 (3440 octets) ... flushed
reading message 2 of 2 (1967 octets) . flushed
You have new mail in /var/spool/mail/matthew
```

By using the `-d` option, along with a time interval (in seconds), you can use Fetchmail in its daemon, or background, mode. The command launches as a background process and retrieves mail from a designated remote server at a specified interval. For more advanced options, see the Fetchmail man page, which is well written and documents all options in detail.

CAUTION

Because the `.fetchmailrc` file contains your mail server password, it should be readable only by you. This means that it should be owned by you and should have permissions no greater than `600`. Fetchmail complains and refuses to start if the `.fetchmailrc` file has permissions greater than this.

Choosing a Mail Delivery Agent

Because of the modular nature of mail handling, it is possible to use multiple applications to process mail and accomplish more than simply deliver it. Getting mail from the storage area and displaying it to the user is the purpose of the MDA. MDA functionality can be found in some of the mail clients (MUAs), which can cause some confusion to those still unfamiliar with the concept of UNIX mail. As an example, the Procmail MDA provides filtering based on rulesets; KMail and Evolution, both MUAs, provide filtering, but the MUAs `pine`, `mutt`, and `Balsa` do not. Some MDAs perform simple sorting, and other MDAs are designed to eliminate unwanted emails, such as spam and viruses.

You would choose an MDA based on what you want to do with your mail. This section presents several MDAs that offer functions you might find useful in your particular situation. If you have simple needs (such as simply organizing mail by rules), one of the MUAs that offers filtering might be better for your needs. Ubuntu provides the Evolution MUA

as the default selection (and it contains some MDA functionality, as previously noted), so try that first and see whether it meets your needs. If not, investigate one of the following MDAs provided by Ubuntu.

Unless otherwise noted, all the MDA software discussed here is provided in the Ubuntu repositories. Chapter 9, “Managing Software,” details the general installation of any software.

Procmail

As a tool for advanced users, the Procmail application acts as a filter for email as it is retrieved from a mail server. It uses rulesets (known as *recipes*) as it reads each email message. No default configuration is provided; you must manually create a `~/.procmail` file for each user, or users can create their own.

There is no system-wide default configuration file. The creation of the rulesets is not trivial and requires an understanding of the use of regular expressions that is beyond the scope of this chapter. Ubuntu does provide three examples of the files in `/usr/share/doc/procmail/examples`, as well as a fully commented example in the `/usr/share/doc/procmail` directory, which also contains a README and FAQ. You can find details for the rulesets in the man page for Procmail and in the man pages for `procmailrc`, `procmailsc`, and `procmailcx`, which contain examples of Procmail recipes.

Spamassassin

If you have used email for any length of time, you have likely been subjected to *spam*, unwanted email sent to thousands of people at the same time. Ubuntu provides an MDA named *Spamassassin* to assist you in reducing and eliminating unwanted emails. Easily integrated with Procmail and Sendmail, it can be configured for both system-wide and individual use. It uses a combination of rulesets and blacklists (Internet domains known to mail spam).

Enabling Spamassassin is simple. You must first have installed and configured Procmail. The README file in `/usr/share/doc/spamassassin` provides details on configuring the `.procmail` file to process mail through Spamassassin. Spamassassin tags probable spam with a unique header; you can then have Procmail filter the mail in any manner you choose. One interesting use of Spamassassin is to tag email received at special email accounts established solely for the purpose of attracting spam. This information is then shared with the Spamassassin site, where these “spam trap”-generated hits help the authors fine-tune the rulesets.

Squirrelmail

Perhaps you do not want to read your mail in an MUA. If you use your web browser often, it might make sense to read and send your mail via a web interface, such as the ones used by Gmail, Hotmail, or Yahoo! Mail. Ubuntu provides Squirrelmail for just that purpose. Squirrelmail, which is written in PHP, supports IMAP and SMTP. It supports MIME attachments and an address book and folders for segregating email.

You must configure your web server to work with PHP 4. You can find detailed installation instructions in `/usr/share/doc/squirrelmail/INSTALL`. After you configure Squirrelmail, point your web browser to the default install location, `www.yourdomain.com/squirrelmail/`, to read and send email.

Virus Scanners

Although the currently held belief is that Linux is immune to email viruses targeted at Microsoft Outlook users, it certainly makes no sense for UNIX mail servers to permit infected email to be sent through them. Although Ubuntu does not provide a virus scanner by default, some of the popular scanners are available in the Ubuntu repositories. Take a look at ClamAV as a popular example.

Autoresponders

Autoresponders automatically generate replies to received messages; they are commonly used to notify others that the recipient is out of the office. Mercifully, Ubuntu does not include one by default, but you can find and install an autoresponder like `vacation` or `gnarwl` from the Ubuntu software repositories. If you are subscribed to a mailing list, be aware that automatic responses from your account can be very annoying to others on the list. Please unsubscribe from mail lists before you leave the office with your autoresponder activated.

References

- ▶ **www.sendmail.org**—The Sendmail home page, where you can find configuration information and FAQs regarding the Sendmail MTA
- ▶ **www.postfix.org**—The Postfix home page, where you can find documentation and sample configurations
- ▶ **https://help.ubuntu.com/community/Postfix**—Ubuntu community documentation for Postfix
- ▶ **www.qmail.org**—The home page for the Qmail MTA, which contains documentation and links to other resources on Qmail
- ▶ **https://help.ubuntu.com/community/ClamAV**—Ubuntu community documentation for ClamAV
- ▶ **www.rfc-editor.org**—A repository of *Requests for Comments (RFCs)*, which define the technical “rules” of modern computer usage
- ▶ **www.procmail.org**—The Procmail home page
- ▶ **Sendmail by Brian Costales, Claus Assmann, George Jansen, and Gregory Neil Shapiro**—The de facto standard guide for everything Sendmail, loaded with more than 1,000 pages, which gives you an idea of how complicated Sendmail really is
- ▶ **Postfix by Richard Blum**—An excellent book that covers the Postfix MTA
- ▶ **Postfix: The Definitive Guide by Kyle D. Dent**—Another excellent resource for Postfix
- ▶ **Running Qmail by Richard Blum**—Similar to the Postfix book by Blum except that it covers the Qmail MTA

CHAPTER 35

Proxying, Reverse Proxying, and Virtual Private Networks (VPNs)

You can never have enough of two things in this world: time and bandwidth. Ubuntu comes with a proxy server, Squid, that enables you to cache web traffic on your server so that websites load faster and users consume less bandwidth. Sometimes proxy servers are recommended for security and privacy, but a virtual private network (VPN) is an even better option if security and privacy are your main concerns. The last section of this chapter is about VPNs. Both proxy servers and VPNs have the interesting side effect that when they are in use, everything that your computer connects to—say a website—assumes that the IP address of the proxy or VPN server is your IP address.

What Is a Proxy Server?

A *proxy server* lies between client machines—the desktops in your company—and the Internet. As clients request websites, they do not connect directly to the Web and send the HTTP request. Instead, they connect to the local proxy server. The proxy then forwards their requests on to the Web, retrieves the result, and hands it back to the client. At its simplest, a proxy server really is just an extra layer between client and server, so why bother?

The three main reasons for deploying a proxy server are as follows:

- ▶ **Content control**—You want to prevent access to certain types of content.

IN THIS CHAPTER

- ▶ What Is a Proxy Server?
- ▶ Installing Squid
- ▶ Configuring Clients
- ▶ Access Control Lists
- ▶ Specifying Client IP Addresses
- ▶ Sample Configurations
- ▶ Virtual Private Networks (VPNs)
- ▶ References

- ▶ **Speed**—You want to cache common sites to make the most of your bandwidth.
- ▶ **Security**—You want to monitor what people are doing.

Squid accomplishes these things and more.

Installing Squid

You can easily install Squid as usual from the Ubuntu software repositories, where it is called `squid`. After Squid is installed, it is automatically enabled for each boot. You can check that it is by running `ps aux | grep squid` when the machine boots. If you see nothing there, run `sudo systemctl start squid`.

Configuring Clients

Before you configure your new Squid server, set up the local web browser to use it for its web access. Doing so enables you to test your rules as you are working with the configuration file.

To configure Firefox, while Firefox is running in the foreground, select Preferences from the Edit menu from the top panel of the Ubuntu desktop. In the dialog that appears, select the Advanced settings using the icon in the top row, and within Advanced, select the Network tab. Then click the Settings button next to Configure How Firefox Connects to the Internet and select the Manual Proxy Configuration option. Check the box beneath it, labeled Use the Same Proxy for All Protocols. Enter `127.0.0.1` in the HTTP Proxy box and `3128` as the port number. See Figure 35.1 for how this should look. If you are configuring a remote client, specify the IP address of the Squid server rather than `127.0.0.1`.

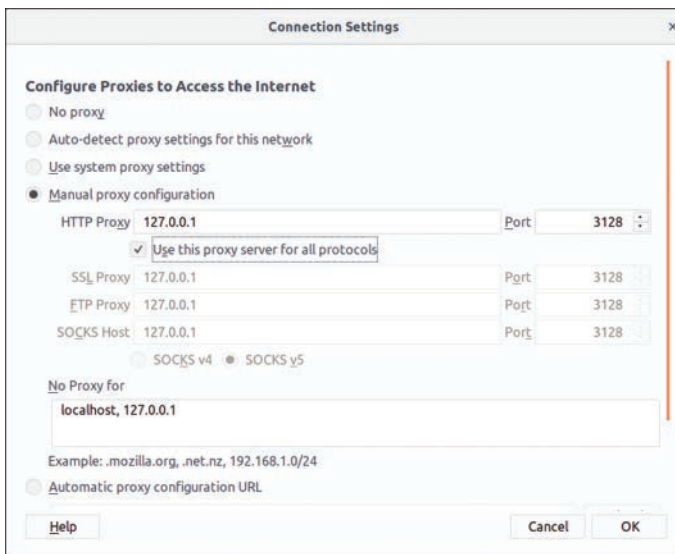


FIGURE 35.1 Setting up Firefox to use `127.0.0.1` routes all its web requests through Squid.

You can similarly configure other web browsers such as Google Chrome, Opera, and so on. The difference is the labels used and menu locations for the options, so you might have to do a little digging to discover where to adjust a specific browser's settings.

Access Control Lists

The main Squid configuration file is `/etc/squid/squid.conf`, and the default Ubuntu configuration file is full of comments to help guide you. The default configuration file allows full access to the local machine but denies the rest of your network. This is a secure place to start; we recommend that you try all the rules on yourself (`localhost`) before rolling them out to other machines.

Open two terminal windows. In the first, change to the directory `/var/log/squid` and run this command:

```
matthew@seymour:~$ sudo tail -f access.log cache.log
```

This reads the last few lines from both files and (thanks to the `-f` flag) follows them so that any changes appear in there. This allows you to watch what Squid is doing as people access it. We refer to this window as the “log window,” and you should keep it open. In the other window (again, with `sudo`), bring up the file `/etc/squid/squid.conf` in your favorite editor. We refer to this window as the “config editor,” and you should keep it open, too.

Search for the string `acl all`; this brings you to the access control section, which is where most of the work needs to be done. You can configure a lot elsewhere, but unless you have unusual requirements, you can leave the defaults in place.

NOTE

The default port for Squid is 3128, but you can change that by editing the `http_port` line. Alternatively, you can have Squid listen on multiple ports by having multiple `http_port` lines; 80, 8000, 8080, and 8888 are all popular ports for proxy servers.

The `acl` lines make up your *access control lists (ACLs)*. The first 16 or so of these lines define the minimum recommended configuration to set up ports to listen to and so on. You can safely ignore these. If you scroll down further (past another short block of comments), you come to the `http_access` lines, which are combined with the `acl` lines to dictate who can do what. You can (and should) mix and match `acl` and `http_access` lines to keep your configuration file easy to read.

Just below the first block of `http_access` lines is a comment like `# INSERT YOUR OWN RULE(S) HERE TO ALLOW ACCESS FROM YOUR CLIENTS`. This is just what you need to do now. First, though, scroll just a few lines further, until you see the following two lines (which are not necessarily next to each other in the actual file):

```
http_access allow localhost
http_access deny all
```

These lines are self-explanatory: The first says, “Allow HTTP access to the local computer but deny everyone else.” This is the default rule, as mentioned earlier. Leave it in place for now and run `service squid start` to start the server with the default settings. If you have not yet configured the local web browser to use your Squid server, do so now so that you can test the default rules.

In your web browser (Firefox is assumed from here on because it is the default in a standard Ubuntu install, but it makes little difference), go to the URL `www.ubuntu.com`. You should see it appear as normal in the browser, but in the log window, you should see a lot of messages scroll by as Squid downloads the site for you and stores it in its cache. This is all allowed because the default configuration allows access to the `localhost`.

Go back to the config editor window and add the following before the last two `http_access` lines:

```
http_access deny localhost
```

So, the last three lines should look like this:

```
http_access deny localhost
http_access allow localhost
http_access deny all
```

Save the file and quit your editor. Then run this command:

```
matthew@seymour:~$ kill -SIGHUP 'cat /var/run/squid.pid'
```

This looks for the *process ID (PID)* of the `squid` daemon and then sends the `SIGHUP` signal to it, which forces it to reread its configuration file while running. You should see a string of messages in the log window as Squid rereads its configuration files. If you now go back to Firefox and enter a new URL, you should see a Squid error page, informing you that you do not have access to the requested site.

The reason you are now blocked from the proxy is because Squid reads its ACL lines in sequence, from top to bottom. If it finds a line that conclusively allows or denies a request, it stops reading and takes the appropriate action. So, in the previous lines, `localhost` is being denied in the first line and then allowed in the second. When Squid sees `localhost` asking for a site, it reads the `deny` line first and immediately sends the error page; it does not even get to the `allow` line. Having a `deny all` line at the bottom is highly recommended so that only those you explicitly allow are able to use the proxy.

Go back to editing the configuration file and remove the `deny localhost` and `allow localhost` lines. This leaves only `deny all`, which blocks everyone (including the `localhost`) from accessing the proxy. Now you are going to add some conditional `allow` statements. You want to allow `localhost` only if it fits certain criteria.

Defining access criteria is done with the `acl` lines, so above the `deny all` line, add this:

```
acl newssites dstdomain news.bbc.co.uk slashdot.org
http_access allow newssites
```

The first line defines an access category called `newssites`, which contains a list of domains (`dstdomain`). The domains are `news.bbc.co.uk` and `slashdot.org`, so the full line reads “Create a new access category called news sites that should filter on domain and contain the two domains listed.” It does *not* say whether access should be granted or denied to that category; that comes in the next line. The line `http_access allow newssites` means “Allow access to the category news sites with no further restrictions.” It is not limited to `localhost`, which means it applies to every computer connecting to the proxy server.

Save the configuration file and rerun the `kill -SIGHUP` line from before to restart Squid; then go back to Firefox and try loading `www.ubuntu.com`. You should see the same error as before because that was not in your `newssites` category. Now try `http://news.bbc.co.uk`, and it should work. However, if you try `www.slashdot.org`, it will *not* work, and you might also have noticed that the images did not appear on the BBC News website either. The problem here is that specifying `slashdot.org` as the website is specific: It means that `http://slashdot.org` will work, whereas `www.slashdot.org` will not. The BBC News site stores its images on the site `http://newsimg.bbc.co.uk`, which is why they do not appear.

Go back to the configuration file and edit the `newssites` ACL to this:

```
acl newssites dstdomain .bbc.co.uk .slashdot.org
```

Putting the period in front of the domains (and in the BBC’s case, taking the `news` off, too) means that Squid will allow any subdomain of the site to work, which is usually what you want. If you want even more vagueness, you can just specify `.com` to match `*.com` addresses.

Moving on, you can also use time conditions for sites. For example, if you want to allow access to the news sites in the evenings, you can set up a time category by using this line:

```
acl freetime time MTWHFAS 18:00-23:59
```

This time, the category is called `freetime` and the condition is `time`, which means you need to specify what time the category should contain. The seven characters following that are the days of the week: Monday, Tuesday, Wednesday, tHursday, Friday, saturday, and Sunday. Thursday and Saturday use capital `H` and `A` so they do not clash with Tuesday and Sunday.

With that category defined, you can change the `http_access` line to include it, like this:

```
http_access allow newssites freetime
```

For Squid to allow access now, it must match both conditions—the request must be for either `*.bbc.co.uk` or `slashdot.org`, and it must be during the time specified. If either condition does not match, the line is not matched, and Squid continues looking for other matching rules beneath it. The times you specify here are inclusive on both sides, which means users in the `freetime` category are able to surf from 18:00:00 until 23:59:59.

You can add as many rules as you like, although you should be careful to try to order them so that they make sense. Keep in mind that all conditions in a line must be matched for the line to be matched. Here is a more complex example:

- ▶ You want a category `newssites` that contains serious websites people need for their work.
- ▶ You want a category `playsites` that contains websites people do not need for their work.
- ▶ You want a category `worktime` that stretches from 09:00 to 18:00.
- ▶ You want a category `freetime` that stretches from 18:00 to 20:00, when the office closes.
- ▶ You want people to be able to access the news sites but not the play sites during working hours.
- ▶ You want people to be able to access both the news sites and the play sites during the free-time hours.

To do all this, you need the following rules:

```
acl newssites dstdomain .bbc.co.uk .slashdot.org
acl playsites dstdomain .tomshardware.com ubuntulinux.org
acl worktime time MTWHF 9:00-18:00
acl freetime time D 18:00-20:00
http_access allow newssites worktime
http_access allow newssites freetime
http_access allow playsites freetime
```

NOTE

You can replace `MTWHF` with `D` as it is equivalent to `MTWHF` and means “all the days of the working week.”

Notice that there are two `http_access` lines for the `newssites` category: one for `worktime` and one for `freetime`. This is because all the conditions must be matched for a line to be matched. Alternatively, you can write this:

```
http_access allow newssites worktime freetime
```

However, if you do this and someone visits `https://news.bbc.co.uk` at 2:30 p.m. (14:30) on a Tuesday, Squid works like this:

- ▶ Is the site in the `newssites` category? Yes, continue.
- ▶ Is the time within the `worktime` category? Yes, continue.
- ▶ Is the time within the `freetime` category? No; do not match rule and continue searching for rules.

It is because of this that two lines are needed for the `worktime` category.

One particularly powerful way to filter requests is with the `url_regex` ACL line. This enables you to specify a regular expression that is checked against each request: If the expression matches the request, the condition matches.

For example, if you want to stop people from downloading Windows executable files, you use this line:

```
acl noexes url_regex -i exe$
```

The dollar sign (\$) means “end of URL,” which means it would match `www.somesite.com/virus.exe` but not `www.executable.com/innocent.html`. The `-i` part means “not case sensitive,” so the rule matches `.exe`, `.Exe`, `.EXE`, and so on. You can use the caret symbol (^) for “start of URL.”

For example, you could stop some pornography sites by using this ACL:

```
acl noporn url_regex -i sex
```

Do not forget to run the `kill -SIGHUP` command each time you make changes to Squid; otherwise, it does not reread your changes.

You can have Squid check your configuration files for errors by running `squid -k parse as root`. If you see no errors, it means your configuration is fine.

NOTE

It is critical that you run the command `kill -SIGHUP` and provide it the PID of your Squid daemon each time you change the configuration; without this, Squid does not reread its configuration files.

Specifying Client IP Addresses

The configuration options so far have been basic, and you can use many more options to enhance the proxying system as you like.

After you have decided which rules work for you locally, it is time to spread them out to other machines. You do so by specifying IP ranges that should be allowed or disallowed access, and you enter them into Squid by using more ACL lines.

If you want to, you can specify all the IP addresses on your network, one per line. However, for networks of more than about 20 people or using *Dynamic Host Control Protocol (DHCP)*, that is more work than necessary. A better solution is to use *classless interdomain routing (CIDR)* notation, which enables you to specify addresses like this:

```
192.0.0.0/8
192.168.0.0/16
192.168.0.0/24
```


Each line has an IP address, followed by a slash and then a number. That last number defines the range of addresses you want covered and refers to the number of bits in an IP address. An IP address is a 32-bit number, typically presented in dotted-quad notation: A.B.C.D. Each of those quads can be between 0 and 255 (although in practice some of them are reserved for special purposes), and each is stored as an 8-bit number.

The first line in the previous code covers IP addresses starting from 192.0.0.0; the /8 part means that the first 8 bits (the first quad, 192) is fixed and the rest is flexible. So, Squid treats that as addresses 192.0.0.0, 192.0.0.1, through to 192.0.0.255, then 192.0.1.0, 192.0.1.1, all the way through to 192.255.255.255.

The second line uses /16, which means Squid allows IP addresses from 192.168.0.0 to 192.168.255.255. The last line has /24, which allows from 192.168.0.0 to 192.168.0.255.

These addresses are placed into Squid using the `src` ACL line, as follows:

```
acl internal_network src 10.0.0.0/24
```

This line creates a category of addresses from 10.0.0.0 to 10.0.0.255. You can combine multiple address groups together, like this:

```
acl internal_network src 10.0.0.0/24 10.0.3.0/24 10.0.5.0/24 192.168.0.1
```

This example allows 10.0.0.0 through 10.0.0.255, then 10.0.3.0 through 10.0.3.255, and finally the single address 192.168.0.1.

Keep in mind that if you are using the local machine and you have the web browser configured to use the proxy at 127.0.0.1, the client IP address will be 127.0.0.1, too. So, make sure you have rules in place for `localhost`.

As with other ACL lines, you need to enable them with appropriate `http_access allow` and `http_access deny` lines.

Sample Configurations

To help you fully understand how Squid access control works, and to give you a head start on developing your own rules, the following are some ACL lines you can try. Each line is preceded with one or more comment lines (starting with a #) explaining what it does:

```
# include the domains news.bbc.co.uk and slashdot.org
# and not newsmg.bbc.co.uk or www.slashdot.org
acl newssites dstdomain news.bbc.co.uk slashdot.org

# include any subdomains or bbc.co.uk or slashdot.org
acl newssites dstdomain .bbc.co.uk .slashdot.org

# only include sites located in Canada
acl canadasites dstdomain .ca
```

```

# only include working hours
acl workhours time MTWHF 9:00-18:00

# only include lunchtimes
acl lunchtimes time MTWHF 13:00-14:00

# only include weekends
acl weekends time AS 00:00-23:59

# include URLs ending in ".zip". Note: the \ is important,
# because "." has a special meaning otherwise
acl zipfiles url_regex -i \.zip$

# include URLs starting with https
acl httpsurls url_regex -i ^https

# include all URLs that match "Hotmail"
url_regex hotmail url_regex -i hotmail

# include three specific IP addresses
acl directors src 10.0.0.14 10.0.0.28 10.0.0.31

# include all IPs from 192.168.0.0 to 192.168.0.255
acl internal src 192.168.0.0/24

# include all IPs from 192.168.0.0 to 192.168.0.255
# and all IPs from 10.0.0.0 to 10.255.255.255
acl internal src 192.168.0.0/24 10.0.0.0/8

```

When you have your ACL lines in place, you can put together appropriate `http_access` lines. For example, you might want to use a multilayered access system so that certain users (for example, company directors) have full access, whereas others are filtered. Here is an example:

```

http_access allow directors
http_access deny hotmail
http_access deny zipfiles
http_access allow internal lunchtimes
http_access deny all

```

Because Squid matches these lines in order, directors will have full, unfiltered access to the Web. If the client IP address is not in the directors list, the two deny lines are processed so that the user cannot download zip files or read online mail at Hotmail. After blocking those two types of requests, the allow rule on line 4 allows internal users to access the Web, as long as they do so only at lunchtime. The last line (which is highly recommended) blocks all other users from the proxy.

When you are done making configuration changes, use `systemd` to start, stop, or restart the squid service, like this:

```
$ sudo systemctl restart squid.service
```

Virtual Private Networks (VPNs)

A *virtual private network*, or *VPN*, creates a way for networks that are otherwise isolated or inaccessible to communicate with one another. Businesses at an enterprise level often use VPNs to keep internal business networks secure while allowing workers to access the internal network from remote locations, such as when an executive is traveling and needs to use a laptop to download and reply to email using an internal business server. The VPN keeps out all traffic except traffic that originates within the network itself or traffic that attempts to connect from the outside using a VPN connection with proper access credentials. This sounds similar to remote access standards already in place in UNIX, Linux, and Ubuntu, but using a VPN takes security to a new level.

There are other types of VPNs in use as well. Not only can VPNs be used to allow remote access to secure internal networks, but they can also be used to allow two networks to connect to one another using a different network in the middle (for example, two networks that each use IPv6 connecting to one another over an IPv4 network using a VPN connection). This is much less common, so we concentrate on the first scenario of a remote user connecting to a secure, internal network. You might be asking how this is different from using a proxy server, as it seems that the VPN is somehow working as an intermediary or a bridge between the remote user and the secure system. It is a little more complicated than that. When a proxy server is in use, it is another layer between the two ends of a connection—an intermediary. When a VPN is in use, it provides direct access between the two ends but via an encrypted tunnel; this is analogous to running a cable directly from one end system to the other, effectively making the remote computer an actual part of the system to which it is connecting. From this moment, the remote system tunnels all of its network traffic through the main system.

Whereas a proxy generally works via a web browser and secures all traffic that passes through the browser, a VPN tunnels all traffic. When using a VPN, the remote computer no longer perceives itself as connected first to the Internet and then to the secure system; rather, it perceives itself as being connected directly to the secure system, with the VPN as its router. The difference is illustrated in Figure 35.2.

Some use an Internet router as a metaphor to help explain how a VPN works. In this analogy, the remote computer connects directly to the VPN, which uses the Internet to connect it to its ultimate host computer, the secure network.

So, why do we care? The differences between proxy servers and a VPN make the most difference when it comes time for implementation. Which will best serve your needs? Here are some facts to help you decide:

- ▶ Proxy servers are usually cheaper and easier to set up. VPNs generally cost more and are more difficult to set up, but after they're set up, they're easy to use and are more secure.
- ▶ A single proxy server can service hundreds or thousands of users, but usually a VPN is designed for one connection that is specific to one remote computer and a secure host. (Yes, exceptions exist, but they are beyond the scope of this introductory material.)
- ▶ Each piece of software that uses a proxy must be set up separately. Using a web browser is the most common way to use a proxy server, but other types of software can also be configured to use proxies. When a VPN is up and running, all Internet software on the computer automatically uses it without additional configuration.

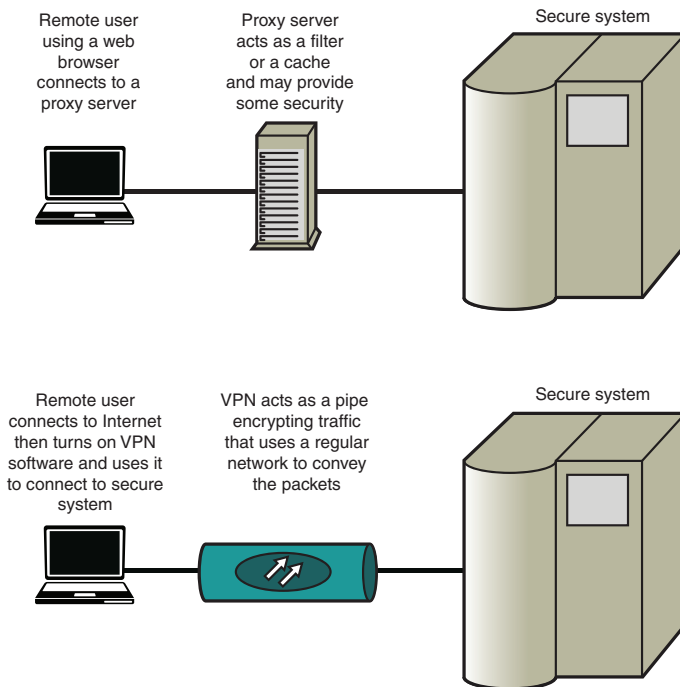


FIGURE 35.2 Comparing a proxy connection to a VPN connection.

Setting Up a VPN Client

The easy part of using a VPN is also the most commonly needed part. Generally, a company sets up VPN servers and then provides access to its secure networks to company employees or clients who use a VPN client installed on their local system, such as a laptop.

Most VPN servers run a protocol that is easily used on Ubuntu, especially from the type of GUI-based system that a typical user would have. You need to check with the

administrator of the VPN network to which you intend to connect to find out which VPN client you will use and also to get your credentials so that you can connect. To manage the VPN client connection in this section, we use the default Network Manager, which is installed by default with Ubuntu and is the default tool for managing all Internet connections in a typical Ubuntu installation.

Install the VPN client software needed for the specific type of VPN server in use by the network to which you will connect. The following table will help you find what you need:

If you need to use	Install this
Cisco Concentrator	<code>network-manager-vpnc</code>
Cisco OpenConnect	<code>network-manager-openconnect</code>
OpenVPN	<code>network-manager-openvpn</code>
PPTP (Microsoft VPN)	<code>network-manager-pptp</code>
strongSwan (for some IPsec VPNs)	<code>network-manager-strongswan</code>

Restart Network Manager to make it aware of the new package(s):

```
matthew@seymour:~$ sudo systemctl restart network-manager
```

Click the Network icon in the top panel of the Unity screen. Click VPN Settings, as shown in Figure 35.3, to be taken to the appropriate location in Settings.

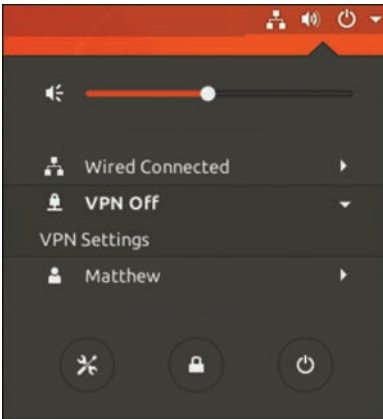


FIGURE 35.3 Network Manager makes configuring a VPN client easy.

In Settings, you need to enter information about your VPN connection. You need to enter things like the gateway IP address of the server, your account username and password, perhaps a group name and group password, and where to find the certificate authority (CA) file. In some cases, you might need to click the Advanced button to enter other details, such as the encryption method, NAT traversal, and more. When you have your information entered, click Save.

To begin using this VPN connection, use the same menu as shown in Figure 36.3, but this time select your VPN connection from the newly created list.

Setting Up a VPN Server

In this section, we use OpenVPN to show how to set up a simple server. Advanced configuration can become quite complex, but it is easy to get started if you require only a basic server.

NEW VPN SOFTWARE OPTION COMING: WIREGUARD

While this book is being written, an open source VPN solution called WireGuard is being merged into the Linux kernel for the 5.6 release. See www.wireguard.com and <https://arstechnica.com/gadgets/2020/01/linus-torvalds-pulled-wireguard-vpn-into-the-5-6-kernel-source-tree/> for more details. Ubuntu 20.04 LTS was originally planned to ship with the 5.5 kernel, but it is likely that the WireGuard code will be backported into that kernel.

You will still need userspace tools and will have to install a package, but WireGuard is smaller than most VPN packages and will be less complex to set up. It is also supposed to be more efficient and performant.

The project is still undergoing testing and is not yet ready for production use, but is likely to become the replacement for OpenVPN (OpenVPN will likely remain available for a long time) during the lifecycle of this edition of *Ubuntu Unleashed*, so it deserves a strong mention. If you want to test WireGuard, install the `wireguard` package from the Ubuntu repositories and see the official quick start documentation to continue at www.wireguard.com/quickstart/.

Install `openvpn` from the Ubuntu software repositories.

Next, you need to create a public key infrastructure (PKI) for OpenVPN. Set up the certificate authority to generate your own certificates and keys:

```
matthew@seymour:~$ sudo mkdir /etc/openvpn/easy-rsa
matthew@seymour:~$ sudo cp -r /usr/share/doc/openvpn/examples/easy-rsa/2.0/* /etc/
openvpn/easy-rsa/
```

Enter your specific details by editing `/etc/openvpn/easy-rsa/vars` and adjusting the following:

```
export KEY_COUNTRY="us"
export KEY_PROVINCE="IA"
export KEY_CITY="Iowa City"
export KEY_ORG="Your Company"
export KEY_EMAIL="yourContact@yourEmailDomain.com"
```

Generate your certificate authority and key:

```
matthew@seymour:~$ cd /etc/openvpn/easy-rsa
matthew@seymour:~$ sudo source vars
matthew@seymour:~$ sudo ./clean-all
```

```
matthew@seymour:~$ sudo ./build-ca
matthew@seymour:~$ sudo cp -r /usr/share/doc/openvpn/examples/easy-rsa/2.0/* /etc/
openvpn/easy-rsa/
```

Generate a certificate and private key for the server, replacing *yourservername* with the name of your server:

```
matthew@seymour:~$ sudo ./buid-key-server yourservername
```

Build the Diffie-Hellman parameters:

```
matthew@seymour:~$ sudo ./build-dh
```

Copy the certificates and keys, replacing *yourservername* with the name of your server:

```
matthew@seymour:~$ cd keys/
matthew@seymour:~$ sudo cp yourservername.crt yourservername.key ca.crt dh1024.pem /
etc/openvpn/
```

You must create a different certificate for each client by using this method. This is because the larger, proprietary VPN vendors distribute their certificates with their server and client software, but you are creating your own. Do this on the server machine for each client, replacing *clientname* with the name of each client system:

```
matthew@seymour:~$ cd /etc/openvpn/easy-rsa/
matthew@seymour:~$ source vars
matthew@seymour:~$ ./build-key clientname
```

Now, copy the following files you just generated to the client for which it was generated. Repeat as needed for each client, replacing *clientname* with the name of each client system:

```
/etc/openvpn/ca.crt
/etc/openvpn/easy-rsa/keys/clientname.crt
/etc/openvpn/easy-rsa/keys/clientname.key
```

Remove the files from the server after they are installed on the client.

Many sample configuration files are included with OpenVPN in `/usr/share/doc/openvpn/examples/sample-config-files/`. You can read through them if you have more complex needs than the simple setup shown here. For this example, you only need the most basic configuration files. Copy and unpack this file:

```
matthew@seymour:~$ sudo cp /usr/share/doc/openvpn/examples/sample-config-files/
server.conf.gz /etc/openvpn
matthew@seymour:~$ sudo gzip -d /etc/openvpn/server.conf.gz
```

Edit `/etc/openvpn/server.conf` to point to and use the certificates and keys you created earlier by changing or adding these lines, replacing `yourservername` with the name of your server and leaving all the other default settings in place:

```
ca ca.crt
cert yourservername.crt
key yourservername.key
```

Start your server:

```
matthew@seymour:~$ sudo systemctl start openvpn-server
```

OpenVPN should create a new networking interface on your computer called `tun0`. To make sure the interface is created, enter the following:

```
matthew@seymour:~$ sudo ifconfig tun0
```

To use your new VPN server with the client described in the previous section, select OpenVPN as the VPN type, enter `yourservername` from this section as the gateway, and set the type to certificates (TLS). Also point the user certificate to use the client certificate you created and moved to the client machine, CA Certificate, to use the credential authority certificate you created and moved to the client machine, and point Private Key to use the private key file you created and moved to the client machine.

References

- ▶ www.squid-cache.org—The home page of the Squid web proxy cache
- ▶ <https://github.com/oskarpearson/squid-users-guide>—The home page of *Squid: A User's Guide*, a free online book about Squid
- ▶ <https://help.ubuntu.com/lts/serverguide/squid.html>—Ubuntu community documentation for setting up Squid
- ▶ *Squid: The Definitive Guide by Duane Wessels*—A practical guide that covers the Squid server in depth, by one of the leading developers on Squid
- ▶ *Web Caching by Duane Wessels*—A theoretical book that discusses how caching is implemented
- ▶ <https://help.ubuntu.com/lts/serverguide/openvpn.html>—The official Ubuntu server documentation for setting up OpenVPN

This page intentionally left blank

CHAPTER 36

Lightweight Directory Access Protocol (LDAP)

Lightweight Directory Access Protocol (LDAP, pronounced “ell-dap”) is one of those technologies that, although hidden, forms part of the core infrastructure in much of enterprise computing. Its job is simple: It stores information about users. However, its power comes from the fact that it can be linked into dozens of other services. LDAP can power login authentication, public key distribution, email routing, and address verification. More recently, it has formed the core of the push toward single sign-on technology.

TIP

Most people find the concept of LDAP easier to grasp when they think of it as a highly specialized form of database server. Behind the scenes, Ubuntu uses a database for storing all its LDAP information; however, LDAP does not offer anything as straightforward as SQL for data manipulation.

OpenLDAP uses Sleepycat Software’s Berkeley DB (BDB), and sticking with that default is highly recommended. However, alternatives exist that might be better for you if you have specific needs.

This chapter looks at a relatively basic installation of an LDAP server, including how to host a company-wide directory service that contains the names and email addresses of employees. LDAP is a client/server system, meaning that an LDAP server hosts the data, and an LDAP client queries it. Ubuntu comes with OpenLDAP as its LDAP server, along with several LDAP-enabled email clients, including Evolution and Mozilla Thunderbird. This chapter covers all three of these applications.

IN THIS CHAPTER

- ▶ Configuring the Server
- ▶ Configuring Clients
- ▶ LDAP Administration
- ▶ References

Because LDAP data is usually available over the Internet—or at least your local network—it is imperative that you make every effort to secure your server. This chapter gives specific instruction on password configuration for OpenLDAP, and we recommend that you follow the instructions closely.

Configuring the Server

If you have been using LDAP for years, you will be aware of its immense power and flexibility. But if you are just trying LDAP for the first time, it will seem like the most broken component you could imagine. LDAP has specific configuration requirements, is vastly lacking in graphical tools, and has a large number of acronyms to remember. On the bright side, all the hard work you put in is worth it because when it works, LDAP improves your networking experience immensely. You should read this entire chapter and understand it before you go any further. Then read the README file in `/etc/ldap/schema` before you begin configuring your server.

The first step in configuring your LDAP server is to install the client and server applications. When you install the `slapd` and `ldap-utils` packages from the Ubuntu repositories, you also install the `libodbc1` package.

By default, Ubuntu configures `slapd` with the minimum options necessary to run the daemon. This chapter shows how to configure everything from that bare-bones installation up to where it will be useful.

Now you need to know the fully qualified domain name (FQDN) of your server. In a moment, you will begin to write/modify some configuration files, and this will be a vital part of that process. The example uses `matthewhelmke.com`. Whenever you see that, change it to your FQDN.

From the FQDN you acquire your domain component, which is the name of your domain, as stored in DNS. This is abbreviated as `dc`. LDAP considers each part of a domain name (separated by dots) to be a domain component. In the example, there are two `dc` items in `matthewhelmke.com`: `matthewhelmke` and `com`.

OpenLDAP uses a separate directory that contains the `cn=config` *directory information tree (DIT)* to configure the `slapd` daemon dynamically. This enables you to modify schema definitions, indexes, and so on without stopping and restarting the service, as was required in earlier versions. You need two files for this configuration: a back end that has only a minimal configuration and a front end that uses a traditional format that is compatible with and accessed by external programs, using established standards.

Creating Your Schema

Start creating your schema by loading some premade schema files. This makes configuration faster and easier by preloading some settings. If you are building an enterprise server, read the official OpenLDAP documentation and start from scratch so that you know precisely what everything on your server is doing and why. For the example in this chapter, load these three files into the directory using the following commands:

```
matthew@seymour:~$ sudo ldapadd -Y EXTERNAL -H ldapi:/// -f /etc/ldap/schema/
```

```
[ic:ccc] cosine.ldif
matthew@seymour:~$ sudo ldapadd -Y EXTERNAL -H ldapi:/// -f /etc/ldap/schema/
[ic:ccc] nis.ldif
matthew@seymour:~$ sudo ldapadd -Y EXTERNAL -H ldapi:/// -f /etc/ldap/schema/
[ic:ccc] inetorgperson.ldif
```

Next, create a file called `backend.matthewhelmke.com.ldif` with these contents:

```
# Load dynamic backend modules
dn: cn=module,cn=config
objectClass: olcModuleList
cn: module
olcModulepath: /usr/lib/ldap
olcModuleload: back_hdb

# Database settings
dn: olcDatabase=hdb,cn=config
objectClass: olcDatabaseConfig
objectClass: olcHdbConfig
olcDatabase: {1}hdb
olcSuffix: dc=matthewhelmke,dc=com
olcDbDirectory: /var/lib/ldap
olcRootDN: cn=admin,dc=matthewhelmke,dc=com
olcRootPW: changeMEtoSOMETHINGbetter
olcDbConfig: set_cachesize 0 2097152 0
olcDbConfig: set_lk_max_objects 1500
olcDbConfig: set_lk_max_locks 1500
olcDbConfig: set_lk_max_lockers 1500
olcDbIndex: objectClass eq
olcLastMod: TRUE
olcDbCheckpoint: 512 30
olcAccess: to attrs=userPassword by dn="cn=admin,dc=matthewhelmke,dc=com" write by
anonymous auth by self write by * none
olcAccess: to attrs=shadowLastChange by self write by * read
olcAccess: to dn.base="" by * read
olcAccess: to * by dn="cn=admin,dc=matthewhelmke,dc=com" write by * read
```

Make sure you change all instances of `matthewhelmke` and `com` to fit your FQDN and change the entry for `olcRootPW` to a more secure password of your choosing. Then add the new file to the directory, which you do as follows, assuming that you are entering this command from the directory where the file was created:

```
matthew@seymour:~$ sudo ldapadd -Y EXTERNAL -H ldapi:/// -f backend.example.com.
ldif
```

Populating Your Directory

The back end is ready. Now you need to populate the front-end directory to make this useful. Create another file called `frontend.matthewhelmke.com.ldif` with the following contents:

```
# Create top-level object in domain
dn: dc=matthewhelmke,dc=com
objectClass: top
objectClass: dcObject
objectclass: organization
o: Example Organization
dc: Example
description: LDAP Example

# Admin user.
dn: cn=admin,dc=matthewhelmke,dc=com
objectClass: simpleSecurityObject
objectClass: organizationalRole
cn: admin
description: LDAP administrator
userPassword: changeMEtoSOMETHINGbetter

dn: ou=people,dc=example,dc=com
objectClass: organizationalUnit
ou: people

dn: ou=groups,dc=matthewhelmke,dc=com
objectClass: organizationalUnit
ou: groups

dn: uid=john,ou=people,dc=matthewhelmke,dc=com
objectClass: inetOrgPerson
objectClass: posixAccount
objectClass: shadowAccount
uid: matthew
sn: Helmke
givenName: Matthew
cn: Matthew Helmke
displayName: Matthew Helmke
uidNumber: 1000
gidNumber: 10000
userPassword: changeMEtoSOMETHINGbetter
gecos: Matthew Helmke
loginShell: /bin/bash
homeDirectory: /home/matthew
shadowExpire: -1
```

```

shadowFlag: 0
shadowWarning: 7
shadowMin: 8
shadowMax: 999999
shadowLastChange: 10877
mail: matthew@matthewhelmke.com
postalCode: 85711
l: Tucson
o: Example
mobile: +1 (520) xxx-xxxx
homePhone: +1 (520) xxx-xxxx
title: System Administrator
postalAddress: I'm not putting it in the book.
initials: MH

dn: cn=example,ou=groups,dc=example,dc=com
objectClass: posixGroup
cn: example
gidNumber: 10000

```

Remember to change the details to fit your information. Then add this file to the LDAP directory:

```

matthew@seymour:~$ sudo ldapadd -x -D cn=admin,dc=example,dc=com -W -f frontend.
example.com.ldif

```

To check that your content has been added to the LDAP directory correctly, you can use `ldapsearch`, as follows:

```

matthew@seymour:~$ ldapsearch -xLLL -b "dc=example,dc=com" uid=john sn givenName cn

dn: uid=matthew,ou=people,dc=matthewhelmke,dc=com
cn: Matthew Helmke
sn: Helmke
givenName: Matthew

```

In this example, `dn` stands for distinguished name, `uid` refers to user identification, `ou` is the organizational unit, `dc` represents domain component, `cn` is common name, `sn` is the family or surname, and many cultures know `givenName` as your first name.

When you use LDAP, you can organize your data in many ways. You can use a number of currently existing schemas, such as in the previous example using the LDIF files you loaded at the start, or you can write your own. The `/etc/ldap/schemas` directory has many fine examples in the files with a `.schema` suffix and a few that have been converted to *LDAP Data Interchange Format (LDIF)*. To be used with LDAP, a file must be an LDIF file (with the `.ldif` filename extension). You can convert one of the sample schemas or create your own schema.

Configuring Clients

Although Ubuntu comes with a selection of email clients, there is not enough room here to cover them all. The two most frequently used clients are Thunderbird, the default, and Evolution. Both are powerful messaging solutions, and both work well with LDAP. Of the two, Thunderbird seems to be the easier to configure and, as the default, is the one we choose to show here.

To enable Thunderbird to use LDAP, go to the menu, click Preferences, and then select Composition from the tabs along the top.

From the Addressing subtab, check the Directory Server box and click the Edit Directories button to its right. In the dialog box that appears, click Add to add a new directory. You can give it any name you want because this is merely for display purposes. As shown in Figure 36.1, set the Hostname field to be the IP address of your LDAP server (or 127.0.0.1 if you are working on the server). Set the Base DN field to the DN for your address book (for instance, `ou=People,dc=matthewhelmke,dc=com`) and leave Port Number set to 389. Click OK three times to get back to the main interface.

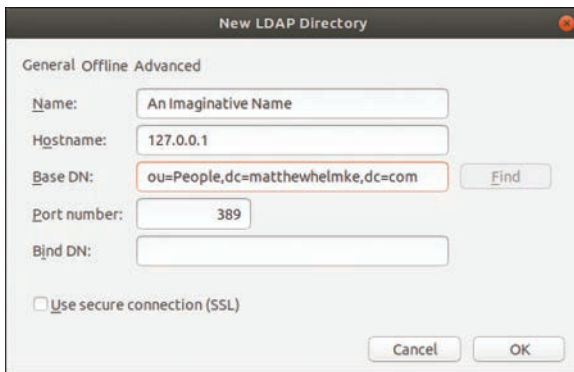


FIGURE 36.1 Thunderbird’s options allow you to download the LDAP directory for offline use.

LDAP Administration

After you have your LDAP server and clients set up, they require little maintenance until something changes externally. Specifically, if someone in your directory changes jobs, changes her phone number, gets married (changing her last name [surname]), quits, or experiences some other change, you need to be able to update your directory to reflect the change.

You installed some useful utilities with the `ldap-utils` package earlier:

- ▶ **ldapsearch**—Opens a connection to an LDAP server and searches its directory for requested information
- ▶ **ldapmodify**—Opens a connection to an LDAP server and allows you to add or modify entries

- ▶ **ldapadd**—Opens a connection to an LDAP server and allows you to add an entry
- ▶ **ldapdelete**—Opens a connection to an LDAP server and allows you to delete one or more entries

Each of these utilities requires administration privileges, so use `sudo` with them all. None of these utilities are simple to use, but they all come with moderate amounts of documentation in their man pages.

A much smarter option is to use `phpLDAPadmin`, which is an LDAP administration tool that enables you to add and modify entries entirely through your web browser. The program is available in the Ubuntu software repositories as `phpldapadmin`.

Starting, stopping, or restarting the `slapd` daemon is done in the usual way:

```
sudo systemctl start/stop/restart slapd
```

References

- ▶ **www.openldap.org**—The home page of the OpenLDAP project, where you can download the latest version of the software and meet other users
- ▶ **<https://ldap.perl.org>**—The home of the Perl library for interacting with LDAP, which provides comprehensive documentation to get you started
- ▶ **<https://help.ubuntu.com/lts/serverguide/openldap-server.html>**—Official Ubuntu Server documentation for OpenLDAP
- ▶ **<https://phpldapadmin.sourceforge.net>**—The official documentation for `phpLDAPadmin`
- ▶ ***LDAP System Administration by Gerald Carter***—The definitive book on LDAP and an absolute must for the bookshelf of any Linux LDAP administrator
- ▶ ***LDAP Directories Explained by Brian Arkills***—More general reading, with a much stronger focus on the Microsoft Active Directory LDAP implementation

This page intentionally left blank

CHAPTER 37

Name Serving with the Domain Name System (DNS)

Humans enjoy naming things. We name our kids, our pets, and often inanimate objects as well. Cars, boats, and computers are not immune. Networked computers are very often named. Remembering names is easier for most of us than remembering numbers. When we must remember numbers, we can; I can still remember the phone number of the house I lived in when I was a kid. However, when we can use a name instead, we tend to prefer to do so; today I can't remember my daughter's cell phone number, because it is programmed into my phone and I just select her picture in the directory to call her.

If you are working in a data center, especially one with a large number of servers networked together, you won't easily remember the IP addresses of all the systems. Besides, it is fun to pick a theme and name everything on the network using that theme: varieties of apples, characters from your favorite science fiction or fantasy world, brands of guitars you wish you owned. It's much more fun and easier to say, "Hey, Liz, can you go check on sontaran for me? It isn't responding to my ping requests," than to say, "Can you check on the server at IP 192.168.2.46?"

Mnemonic devices like these are useful, but we must map the names we give to the numbers the systems actually use. Mapping is just matching, connecting the words with the numbers in a way that allows a human to request `https://google.com` and get the information stored at `https://74.125.224.72`.

IN THIS CHAPTER

- ▶ Understanding Domain Names
- ▶ Setting Up a DNS Server with BIND
- ▶ References

The Domain Name System (DNS) converts human-readable names given to networked machines to the IP addresses that machines use. (See Chapter 18, “Networking,” for more on IP addresses.) DNS works in the other direction as well, taking IP addresses and giving names.

When you set up a DNS server using the BIND DNS server software discussed in this chapter, you gain the power to set the name/IP details for domains you own, either domains that are internal on your network or Internet domains you have purchased through a registrar like Namecheap.com or GoDaddy. But these registrars run DNS servers and allow you to perform this setup through their lovely web GUIs. Why would anyone want to do this themselves? There are several possible reasons:

- ▶ **You have an internal network, a local area network (LAN) that is not accessible from outside that network**—Your internal network assigns and uses private IP addresses, but you want to make things easier for your LAN users by assigning names to the various systems. These domain names will not be assigned by a domain name registrar because they are internal only; they won’t include top-level domains like .com or .net at the end and will be usable only within the network.
- ▶ **You want to improve performance by caching**—Most DNS queries are shared by a large number of computers. If those individual computers don’t have to each connect to the Internet and get their domain names resolved by your Internet service provider’s DNS server (or another public DNS server) but can instead get that information from a server on the local network, it could speed things up. This isn’t an issue for most of us, but for large networks, it can be a huge benefit.
- ▶ **You want to ban access to a harmful domain**—Maybe an email came in to your company’s employees with a link to a specific website that is distributing viruses and you want to prevent naïve internal network users from accessing that website. You can change the IP address that is served for this domain name so that it instead forwards users to an HTML page you write up describing the problem. Your change only affects computers on your internal network, so this is not something that will make people outside your company angry or that they will even know is happening.
- ▶ **You run a local network at home**—You want to make your life a little easier when it comes time to log in to each and perform updates, so you want to give a name to each computer and allow access to each computer using a pet name for each one instead of using the IP address on your network.

No doubt we could come up with other examples. Most of us will not need to set up or run a DNS server, but you are the type of person who reads books like *Ubuntu Unleashed*, so you probably like to tinker with technology. It is likely that most people reading this just want to play with something new and have some fun. This chapter will get you started.

Understanding Domain Names

Simply put, a *domain name* is a string of characters that is used to represent an IP address. Domain names are intended to be easier to remember than strings of numbers. Generally,

they are set up as a set of letters separated by dots. Each of these sets of letters is called a *label*. The label on the far right denotes the widest group, or top-level domain (TLD). Each label to the left is a subdomain of the one to the right of it. `news.google.com` is an example.

Here we see three labels:

- ▶ The top-level domain here is `com`. The original set of top-level domains is `com`, `edu`, `gov`, `mil`, `net`, and `org`. Other top-level domains include country-code TLDs such as `uk` or `in`, sponsored TLDs like `aero` for the Société Internationale de Télécommunications Aéronautiques, and geographic TLDs like `asia`. Others also exist. All TLDs are controlled by a group called ICANN, the Internet Corporation for Assigned Names and Numbers, with which domain names must be registered to work on the Internet (see www.icann.org). The TLD generally defines, with varying firmness, the content permitted on subdomains; some TLDs are far more controlled than others.
- ▶ The first subdomain, in this case `google`, is called the *root zone*. The root zone is required to create a fully qualified domain name (FQDN), which is the combination of the root zone and the TLD. When you register a domain name with a registrar, so you can put a website on the Internet, you register a root zone for a TLD, as in `matthewhelmke.com` OR `matthewhelmke.net`. The complete FQDN is required for DNS to work across the Internet.
- ▶ After the first subdomain, others may be created, such as `news` in our example. These are optional. Some standard ones are `www`, `mail`, and `ftp`, which each denote specific uses for defined subdomains. They are not required, but if you want to host more than one thing using a FQDN, you must define and use subdomains. Subdomains are defined in a DNS server in the same manner as TLDs.

When a full set consisting of TLD and all needed subdomains is put together, this is called a *hostname*. On a local network, this could be a single word. On the Internet, it requires at least a TLD and root zone and often includes at least one other subdomain, as in `www.icann.org`. Another way to define a hostname is any domain name that is associated with one or more IP addresses. The “or more” is often used in load balancing between one or more machines. You can have DNS cycle from one IP address to another each time the hostname is requested.

DNS Servers

A DNS server receives a request or query and responds by resolving that query and returning the information mapped to it, if it exists. There is not one canonical DNS server for the entire Internet. Instead, when a domain is registered, that registration is listed on a series of ICANN-run servers scattered around the world. Contained in that registration is information about where the authoritative DNS information is stored for that domain. This is usually, but not necessarily, a DNS server or set of DNS servers operated by the domain name registrar.

DNS requests work in a hierarchical fashion. Here is an example:

1. A full request comes in to your Internet service provider’s DNS server. If the DNS server knows the information, it responds.

2. If the DNS server recognized the root name but not a subdomain, it submits a request itself to a known server with information about that root name and, if successful, it returns the information it receives.
3. If the DNS server cannot find any cached information about the hostname, it contacts an ICANN server using the TLD. The ICANN server can then respond with the information it has, which at a minimum is the registered IP address that correlates to the hostname, if it exists.

This hierarchy causes some DNS requests to take longer than others. Sometimes the search involves multiple DNS servers across a wider and wider set of data. It also means that one server going down is not catastrophic; instead, the information can be searched for and found elsewhere.

DNS Records

All DNS records are placed in a zone file (discussed further later in this chapter, in the section “Setting Up a DNS Server with BIND”). There are several ways to define the IP address/hostname. Some of these DNS records are optional. Many can have multiple entries. The most critical and commonly used ones are listed here, but many more are available, and many more options are also available than are listed here. See https://en.wikipedia.org/wiki/List_of_DNS_record_types for more information.

A

The A record maps a hostname to a 32-bit IPv4 address, as in this example:

```
example.com IN A 192.0.2.0
```

The hostname comes first. `IN` indicates Internet. `A` indicates that this is an A record. The IPv4 address comes last.

AAAA

The AAAA record maps a hostname to a 128-bit IPv6 address, as in this example:

```
example.com AAAA 2001:db8::/32
```

The hostname comes first, followed by four `As` because 128-bit IPv6 addresses are four times larger than 32-bit IPv4 addresses. (See Chapter 18 for more about IPv4 and IPv6 addresses.) The IPv6 address comes last.

CNAME

The CNAME record maps one or more aliases to the canonical name of a machine. The aliased domain receives all the subdomains and DNS records of the original, canonical name it is matched with. For example, if you have a machine named `weirdname.example.com` that is set up to be an email server, but you want to use a different URL, you can set up an alias so it also uses `mail.example.com`, as in this example:

```
mail.example.com CNAME weirdname.example.com
```

The alias comes first. `CNAME` indicates that the alias is being mapped to the canonical name that comes next. The server that has an `A` record is listed last. `CNAME` records never point to IP addresses, only to other domain names that are already defined using `A` or `AAAA` records.

One neat trick is that you can create a `CNAME` record for every subdomain on a machine that is running multiple services, such as `ftp.example.com`, `www.example.com`, and `mail.example.com`, and have every one of them point to the same `A` or `AAAA` record for the server with the name `example.com`. That `A` or `AAAA` record points to the IP address, so if you ever need to change the IP address, such as when you move your server or establish a new server, you only have to change the IP record in one place.

MX

An `MX` record maps a domain name to a list of mail servers for that domain. (`MX` comes from “mail exchange.”) If you do not use a domain for email, this record is not needed. Here is an example of an `MX` record:

```
example.com. 14400 IN MX 0 mail.example.com.
```

First, notice the dot after each of the two domain names. These are vital, and omitting them causes email to be misrouted.

The example says that any email coming in to an address `@example.com` (leftmost field) should be routed to the server at `example.com` (rightmost field). The DNS `A` or `AAAA` record for `mail.example.com` or a `CNAME` record sets the IP address of that mail server. The `14400` defines how often, in seconds, to update the DNS entry in any other server that has cached this record; `14400` is 4 hours and is standard. `MX` defines this as a mail exchange record.

The `0` indicates preference because you can list more than one `MX` server. Smaller numbers are preferred over larger numbers. Setting multiple servers to the same value means that a random server with that value is to be used. If there is only one, use `0`. A multiple-server `MX` record looks like this:

```
example.com. 14400 IN MX 10 mail.example.com.
example.com. 14400 IN MX 20 mail2.example.com.
```

NS

An `NS` records maps a domain name to a list of DNS servers that are authoritative for that domain. It is most used at domain name registrars and tells any query that comes in to the domain name registrar where to look for the complete DNS record. For example, a request from an ICANN server would look first at the domain name registrar, which would then look wherever it says in the `NS` record, if the complete DNS is not hosted here. Here is an example:

```
example.com. IN NS ns1.domainregistrationcompany.com.
```

The domain comes first, again with a dot at the end. This is followed by `IN` for Internet and `NS` for name server. At the end, the name server is listed, again with a dot at the end.

SOA

A start of authority, or SOA, record specifies the DNS server that provides authoritative information about a domain, the email address of the domain administrator, the domain serial number, and configuration for timers related to refreshing the zone. This is a vital part of the zone file. Here is an example:

```
; name TTL class rr Nameserver email-address
example.com. 14400 IN SOA ns.mydomainnameserver.com. root.ns.mydomainnameserver.com. (
2018080600 ; serial number
86000 ; refresh rate in seconds
7200 ; update retry in seconds
3600000 ; expiration in seconds
600 ; minimum in seconds )
```

In this file, a semicolon denotes the beginning of a comment, so everything after it is ignored by the system and is intended only for humans reading the file. The first line reminds you how to format the next line. Everything else in the file is contained in the parentheses, the first of which must be on the first line.

This example uses `example.com` as the domain. Remember the dot at the end.

`14400` is the TTL, or time in seconds that you are allowing the record to be cached by other servers before they have to submit a query again. If set to `0`, it is not permitted to be cached at all.

`IN` is the class or type of the record—in this case, Internet. No other options are in use any more.

`SOA` denotes this is a start of authority record.

This example uses `ns.mydomainnameserver.com` as the sample domain of the DNS server. Replace this with your DNS server's domain and remember the dot at the end.

The email address of the domain name administrator is a little confusing because you expect an `@` symbol but none exists. In the preceding example, `root.ns.mydomainnameserver.com` means `root@ns.mydomainnameserver.com`. Replace this with the email address for your admin, using a dot instead of the `@` symbol. Remember the dot at the end.

The serial number is a revision numbering system. It is changed every time the file is changed. Convention is to use `YYYYMMDDnn`, where `YYYY` is year, `MM` is month, `DD` is day, and `nn` is an extra number to allow you to increment when multiple edits and saves occur in one day. For example, `2018080600` is the first edit on August 6, 2018.

The refresh rate sets the time, in seconds, when the slave DNS server will refresh from the master DNS server.

The retry rate sets how long to wait after a failed refresh before making another attempt.

The expiration sets how long to keep a zone file cached.

Minimum is the default time that slave servers should cache the zone file. If your DNS record changes frequently, you want to set this to a low number, like 12 hours or so. If infrequently, then every one to five days is a good balance between keeping updated and keeping requests served quickly.

TXT

You can put any text you like in a TXT record. This record is most commonly used to implement the Sender Policy Framework (SPF), which is an email validation system designed to help detect and prevent email spam by detecting email spoofing. With *spoofing*, email is sent out to look like it originated in one location when it actually originated somewhere else. SPF records must indicate the version identifier for SPF and a default mechanism. Here is an example of SPF info in a TXT record:

```
example.com. TXT "v=spf1 -all"
```

This indicates to use SFP version 1 and that no servers at this domain send email. If your server does not send email, use this to prevent email/web hosting companies from blocking your domain if someone tries to send email pretending to come from your domain.

If your server sends email, you use something like this:

```
example.com. TXT "v=spf1 mx -all"
```

The difference here is that `mx` has been added to indicate that any servers tied to this domain may send email.

Many more variations of SPF records are available. Itemize what you have and then read the specification at www.openspf.org/SPF_Record_Syntax to learn what you need to write.

Setting Up a DNS Server with BIND

The Berkeley Internet Name Domain (BIND) software has been foundational to the Internet since the 1980s. The original BIND was created at the University of California, Berkeley, using grant funding. It was released as free and open source with the BSD (Berkeley Software Distribution) version of UNIX and was quickly adopted as the standard software for DNS. The Internet Systems Consortium now maintains BIND and provides updates and documentation at www.isc.org.

When you create NS records for your domains with your domain name registrar, pointing at `ns1.domainregistrationcompany.com` or whatever domain the registrar gives you, you are most likely pointing to a server running BIND, although other options exist. This section shows how to install and set up a very basic DNS server using BIND.

Start by installing `bind9` from the Ubuntu software repositories.

Next, use your favorite text editor to open `/etc/bind/named.conf.local`, which is a configuration file set up with default settings. This is where you must declare the zones you are setting up and associating with the domain. Zones are domain names that are

referenced in the DNS server. Leave all the default text in the file and add this to the end of the file, after replacing `example.com` with your domain name:

```
# This is the zone definition.
zone "example.com" {
    type master;
    file "/etc/bind/zones/example.com.db";
};
```

Save the file and exit.

Next, open the configuration options file at `/etc/bind/named.conf.options` and modify the section titled `forwarders` by replacing `8.8.8.8` shown in this example with the IP address of your provider's DNS server:

```
forwarders {
    8.8.8.8;
};
```

Save the file and exit.

Now you must create a zone definition file. First, create a directory to hold it:

```
matthew@seymour:~$ sudo mkdir /etc/bind/zones
```

Then create a new file but replace `example.com` in this example with your domain:

```
matthew@seymour:~$ sudo touch /etc/bind/zones/example.com.db
```

This zone definition file will contain all the addresses and machine names that you are hosting. This is where you enter all the DNS records discussed in the previous section of the book. Each record gets its own line. For human readability, it is nice to put an empty line after each one and start each record with a comment, using `//` to begin comment lines. In this example, replace `example.com` with your domain, replace `ns` with the domain for your DNS server, and enter whatever other records you want to enter, modifying as the comments note here:

```
example.com. 14400 IN SOA ns.example.com. admin.ns.example.com. (
2013080600 ; serial number
86000      ; refresh rate in seconds
7200      ; update retry in seconds
3600000   ; expiration in seconds
600       ; minimum in seconds )
```

```
// Replace the following lines as necessary:
// example.com = your domain name
// ns1 = your DNS server name
// mta = your mail server name
example.com.      IN      NS      ns1.example.com.
example.com.      IN      MX      10      mta.example.com.
```

```
// Replace the IP address with your IP addresses.
www          IN      A       192.168.0.2
mta          IN      A       192.168.0.3
ns1         IN      A       192.168.0.1
```

Now restart BIND:

```
matthew@seymour:~$ sudo systemctl restart bind9.service
```

You can also use `start`, `stop`, and so on, as you would with other services using `systemd`.

You should now be all set. To test your DNS server, edit `/etc/resolv.conf` (after making a backup of the original!) and replace the contents with the following, replacing `example.com` with your domain name and `192.168.1.1` with the address of your new DNS server:

```
search example.com
nameserver 192.168.1.1
```

Then test that your DNS server is functioning by using `ping` to see what it returns for `example.com`:

```
matthew@seymour:~$ ping example.com
```

In addition to the `ping` utility just mentioned, `dig` is another excellent tool for reviewing DNS records. You may also find `nslookup` and `whois` useful. Also, `named-checkzone` is included with BIND and may be useful to you.

References

- ▶ **www.bind9.net**—A website with tons of useful information about both DNS and BIND
- ▶ ***DNS and BIND* by Cricket Liu and Paul Albitz**—The canonical guide to understanding both of these technologies

This page intentionally left blank

CHAPTER 38

Using Programming Tools

If you're looking to learn C, C++, or Java programming, this part of the book isn't the right place to start. Unlike with Perl, Python, PHP, or even C#, producing something productive with languages like C, C++, and Java takes more than a little dabbling. This chapter is primarily focused on the tools Ubuntu offers you as a programmer.

Whether you're looking to compile your own code or someone else's, the *GNU Compiler Collection* (`gcc`) is there to help. It understands C, C++, Fortran, Pascal, and dozens of other popular languages, which means you can try your hand at whatever interests you. Ubuntu also ships with hundreds of libraries you can link to, from the GUI toolkits behind GNOME and KDE to XML parsing and game coding. Some libraries use C, others C++, and still others offer support for both, meaning you can choose what you're most comfortable with.

WHY USE C OR C++?

Every language has benefits and shortcomings. Some languages make life easier for the programmer but at the expense of runtime. Languages such as Perl and Python and even Java make it hard for the user to guarantee that memory is fetched sequentially or that it fits in cache, due to things such as checks on the bounds on each access. They are useful languages, but they run more slowly than languages that are harder for the programmer but faster at runtime, such as C or Fortran.

For some programs, such as short shell scripts or quick one-liners in Perl to search text in a file, the difference in runtime speed is negligible. On a desktop computer, it might not matter that your music player is written in Python, and if it seems slow, buying a newer, faster desktop system might be an acceptable solution.

IN THIS CHAPTER

- ▶ Programming in C with Linux
- ▶ Using the C Programming Project Management Tools Provided with Ubuntu
- ▶ Using the GNU C Compiler
- ▶ Graphical Development Tools
- ▶ Programming in Java with Linux
- ▶ Graphical Development Tools
- ▶ Beginning Mobile Development for Android
- ▶ Version Control Systems
- ▶ Continuous Integration and Continuous Delivery and DevOps Tools
- ▶ Canonical-created Tools
- ▶ References

There are some applications, however, where the time needed to run your program can make a big difference. For example, using a slow-to-run language to perform calculations on scientific data, especially if you are doing it on *high-performance computing (HPC)* resources like a supercomputing cluster, is foolish; to take advantage of the platform, it is both time and cost effective to use the fastest language available to you, like C.

This idea was reinforced in a 2011 conversation between Matthew Helmke and Dan Stanzione, then deputy director of the Texas Advanced Computing Center at the University of Texas at Austin. Stanzione said that HPC resources are expensive, so it is often wiser to spend grant money to hire a good C programmer for a year than it is to run a bioinformatics program written in Perl or Python on an HPC system. As he put it, “If your computer costs \$2,000, the programmer’s time is the dominant cost, and that is what drives software development. If your computer costs \$100 million or more, then having a programmer spend an extra month, or year, or decade working on software optimization is well worth it. Toughen up and write in C.”

Programming in C with Linux

C is the programming language most frequently associated with UNIX-like operating systems such as Linux and BSD. Since the 1970s, the bulk of the UNIX operating system and its applications have been written in C. Because the C language doesn’t directly rely on any specific hardware architecture, UNIX was one of the first portable operating systems. In other words, the majority of the code that makes up UNIX doesn’t know and doesn’t care which computer it is actually running on. Machine-specific features are isolated in a few modules within the UNIX kernel, which makes it easy for you to modify them when you are porting to different hardware architectures.

Because C is so important to UNIX and Linux, we use it in the examples in this section. Much of what is discussed here also applies to other languages, perhaps with slight variations for language-specific features.

C is a *compiled* language, which means that your C source code is first analyzed by the *preprocessor* and then translated into assembly language before it’s translated into machine instructions that are appropriate to the target CPU. An assembler then creates a binary, or *object*, file from the machine instructions. Finally, the object file is linked to any required external software support by the *linker*. A C program is stored in a text file that ends with a `.c` extension and always contains at least one routine, or function, such as `main()`, unless the file is an *include* file (with a `.h` extension, also known as a *header* file) containing shared variable definitions or other data or declarations. *Functions* are the commands that perform each step of the task that the C program was written to accomplish.

NOTE

The Linux kernel is mostly written in C, which is why Linux works with so many different CPUs. To learn more about building the Linux kernel from source, see Chapter 22, “Kernel and Module Management.”

C++ is an object-oriented extension to C. Because C++ is a superset of C, C++ compilers compile C programs correctly, and writing non-object-oriented code in C++ is possible. The reverse is not true: C compilers cannot compile C++ code.

C++ extends the capabilities of C by providing the necessary features for object-oriented design and code. C++ also provides some features, such as the capability to associate functions with data structures that do not require the use of class-based object-oriented techniques. For these reasons, the C++ language enables existing UNIX programs to migrate toward the adoption of object orientation over time.

Support for C++ programming is provided by `g++`, which you run with the name `g++` when you are compiling C++ code.

Using the C Programming Project Management Tools Provided with Ubuntu

Ubuntu is replete with tools that make your life as a C/C++ programmer easier. There are tools to create programs (editors), compile programs (`gcc`), create libraries (`ar`), control the source (Git, Subversion, Bazaar), automate builds (`make`), debug programs (`gdb` and `ddd`), and determine where inefficiencies lie (`gprof`).

The following sections introduce some of the programming and project management tools included with Ubuntu. If you have some previous UNIX experience, you will be familiar with most of these programs because they are traditional complements to a programmer’s suite of software.

Building Programs with `make`

You use the `make` command to automatically build and install a C program, and for that use it is an easy tool. If you want to create your own automated builds, however, you need to learn the special syntax that `make` uses; the following sections walk you through a basic `make` setup.

Using Makefiles

The `make` command automatically builds and updates applications by using a makefile. A *makefile* is a text file that contains instructions about which options to pass on to the compiler preprocessor, the compiler, the assembler, and the linker. The makefile also

specifies, among other things, which source code files have to be compiled (and the compiler command line) for a particular code module and which code modules are needed to build the program—a mechanism called *dependency checking*.

The beauty of the `make` command is its flexibility. You can use `make` with a simple makefile, or you can write complex makefiles that contain numerous macros, rules, or commands that work in a single directory or traverse your file system recursively to build programs, update your system, and even function as document management systems. The `make` command works with nearly any program, including text processing systems such as TeX.

You could use `make` to compile, build, and install a software package, using a simple command like this:

```
matthew@seymour:~$ sudo make install
```

You can use the default makefile (usually called `Makefile`, with a capital *M*), or you can use `make`'s `-f` option to specify any makefile, such as `MyMakeFile`, like this:

```
matthew@seymour:~$ sudo make -f MyMakeFile
```

Other options might be available, depending on the contents of your makefile. You might have a source file named `hi.c` and just run `make hi`, where `make` figures out what to do automatically to build the final executable. See `make`'s built-in rules with `make -p`.

Using Macros and Makefile Targets

Using `make` with macros can make a program portable. Macros allow users of other operating systems to easily configure a program build by specifying local values, such as the names and locations, or *pathnames*, of any required software tools. In the following example, macros define the name of the compiler (`CC`), the installer program (`INS`), where the program should be installed (`INSDIR`), where the linker should look for required libraries (`LIBDIR`), the names of required libraries (`LIBS`), a source code file (`SRC`), the intermediate object code file (`OBJS`), and the name of the final program (`PROG`):

```
# a sample makefile for a skeleton program
CC= gcc
INS= install
INSDIR = /usr/local/bin
LIBDIR= -L/usr/X11R6/lib
LIBS= -lXm -lSM -lICE -lXt -lX11
SRC= skel.c
OBJS= skel.o
PROG= skel

skel:  ${OBJS}
       ${CC} -o ${PROG} ${SRC} ${LIBDIR} ${LIBS}

install: ${PROG}
         ${INS} -g root -o root ${PROG} ${INSDIR}
```

NOTE

The indented lines in the previous example are indented with tabs, not spaces. This is important to remember! It is difficult for a person to see the difference, but `make` can tell. If `make` reports confusing errors when you first start building programs under Linux, check your project's makefile for the use of tabs and other proper formatting.

Using the makefile from the preceding example, you can build a program like this:

```
matthew@seymour:~$ sudo make
```

To build a specified component of a makefile, you can use a target definition on the command line. To build just the program, you use `make` with the `skel` target, like this:

```
matthew@seymour:~$ sudo make skel
```

If you make any changes to any element of a target object, such as a source code file, `make` rebuilds the target automatically. This feature is part of the convenience of using `make` to manage a development project. To build and install a program in one step, you can specify the target of `install` like this:

```
matthew@seymour:~$ sudo make install
```

Larger software projects might have a number of traditional targets in the makefile, such as the following:

- ▶ **test**—To run specific tests on the final software
- ▶ **man**—To process an include or a `troff` document with the `man` macros
- ▶ **clean**—To delete any remaining object files
- ▶ **archive**—To clean up, archive, and compress the entire source code tree
- ▶ **bugreport**—To automatically collect and then mail a copy of the build or error logs

Large applications can require hundreds of source code files. Compiling and linking these applications can be a complex and error-prone task. The `make` utility helps you organize the process of building the executable form of a complex application from many source files.

Using the `autoconf` Utility to Configure Code

The `make` command is only one of several programming automation utilities included with Ubuntu. There are others, such as `pmake` (which causes a parallel make); `imake` (which is a dependency-driven makefile generator that is used for building X11 clients); `automake`; and one of the newer tools, `autoconf`, which builds shell scripts that can be used to configure program source code packages.

Building many software packages for Linux that are distributed in source form requires the use of GNU's `autoconf` utility. This program builds an executable shell script named

`configure` that, when executed, automatically examines and tailors a client's build from source according to software resources, or *dependencies* (such as programming tools, libraries, and associated utilities) that are installed on the target host (your Linux system).

Many Linux commands and graphical clients for X downloaded in source code form include `configure` scripts. To configure the source package, build the software, and then install the new program, the root user might use the script like this (after uncompressing the source and navigating into the resulting build directory):

```
matthew@seymour:~$ ./configure ; make ; sudo make install
```

The `autoconf` program uses a file named `configure.in` that contains a basic *ruleset*, or set of macros. The `configure.in` file is created with the `autoscan` command. Building a properly executing `configure` script also requires a template for the makefile, named `Makefile.in`. Although creating the dependency-checking `configure` script can be done manually, you can easily overcome any complex dependencies by using a graphical project development tool such as KDE's KDevelop or GNOME's Glade. (See the "Graphical Development Tools" section, later in this chapter, for more information.)

Debugging Tools

Debugging is both a science and an art. Sometimes, the simplest tool—the code listing—is the best debugging tool. At other times, however, you need to use other debugging tools, such as `splint`, `gprof`, and `gdb`.

Using `splint` to Check Source Code

The `splint` command is similar to the traditional UNIX `lint` command: It statically examines source code for possible problems, and it also has many additional features. Even if your C code meets the standards for C and compiles cleanly, it might still contain errors. `splint` performs many types of checks and can provide extensive error information. For example, this simple program might compile cleanly and may even run:

```
matthew@seymour:~$ gcc -o tux tux.c
matthew@seymour:~$ ./tux
```

But the `splint` command might point out some serious problems with the source:

```
matthew@seymour:~$ splint tux.c
Splint 3.1.2 -- 29 Apr 2009
```

```
tux.c: (in function main)
tux.c:2:19: Return value (type int) ignored: putchar(t[++j]) -...
    Result returned by function call is not used. If this is intended, can cast
    result to (void) to eliminate message. (Use -retvalint to inhibit warning)
Finished checking -- 1 code warning
```

You can use the `splint` command's `-strict` option, like this, to get a more verbose report:

```
matthew@seymour:~$ splint -strict tux.c
```

`gcc` also supports diagnostics through the use of extensive warnings (through the `-Wall` and `-pedantic` options):

```
matthew@seymour:~$ gcc -Wall tux.c
tux.c:1: warning: return type defaults to 'int'
tux.c: In function 'main':
tux.c:2: warning: implicit declaration of function 'putchar'
```

Using `gprof` to Track Function Time

You use the `gprof` (profile) command to study how a program is spending its time. If a program is compiled and linked with `-p` as a flag, a `mon.out` file is created when it executes, with data on how often each function is called and how much time is spent in each function. `gprof` parses and displays this data. An analysis of the output generated by `gprof` helps you determine where performance bottlenecks occur. Using an optimizing compiler can speed up a program, but taking the time to use `gprof`'s analysis and revising bottleneck functions significantly improves program performance.

Doing Symbolic Debugging with `gdb`

The `gdb` tool is a symbolic debugger. When you compile a program with the `-g` flag, the symbol tables are retained, and you can use a symbolic debugger to track program bugs. The basic technique is to invoke `gdb` after a *core dump* (which involves taking a snapshot of the memory used by a program that has crashed) and get a stack trace. The stack trace indicates the source line where the core dump occurred and the functions that were called to reach that line. Often, this is enough to identify a problem. It isn't the limit of `gdb`, though.

`gdb` also provides an environment for debugging programs interactively. Invoking `gdb` with a program enables you to set breakpoints, examine the values of variables, and monitor variables. If you suspect a problem near a line of code, you can set a breakpoint at that line and run `gdb`. When the line is reached, execution is interrupted. You can check variable values, examine the stack trace, and observe the program's environment. You can single-step through the program to check values. You can resume execution at any point. By using breakpoints, you can discover many bugs in code.

A graphical X Window interface to `gdb` is called the *Data Display Debugger*, or `ddd`.

Using the GNU C Compiler

If you elected to install the development tools package when you installed Ubuntu (or perhaps later on, using `synaptic`), you should have the *GNU C compiler* (`gcc`). Many different options are available for the GNU C compiler, and many of them are similar to those of the C and C++ compilers that are available on other UNIX systems. Look at the man page or information file for `gcc` for a full list of options and descriptions.

NOTE

The GNU C compiler is a part of the GNU Compiler Collection, which also includes compilers for several other languages.

When you build a C program using `gcc`, the compilation process takes place in several steps:

1. First, the C preprocessor parses the file. To do so, it sequentially reads the lines, includes header files, and performs macro replacement.
2. The compiler parses the modified code to determine whether the correct syntax is used. In the process, it builds a symbol table and creates an intermediate object format. Most symbols have specific memory addresses assigned, although symbols defined in other modules, such as external variables, do not.
3. In the last compilation stage, linking, the GNU C compiler ties together different files and libraries and then links the files by resolving the symbols that had not previously been resolved.

NOTE

Most C programs compile with a C++ compiler if you follow strict ANSI rules. For example, you can compile the standard `hello.c` program (everyone's first program) with the GNU C++ compiler. Typically, you name the file something like `hello.cc`, `hello.C`, `hello.c++`, or `hello.cxx`. The GNU C++ compiler accepts any of these names.

Programming in Java with Linux

The Java programming language was originally developed by Sun Microsystems in the 1990s. The goals were to implement a virtual machine and a language that had a familiar C-like syntax, but that was simpler and with a promise to “write once, run everywhere” by creating a free Java runtime for all popular platforms. They succeeded.

In May 2007, Sun Microsystems released almost all of its Java technologies with an open source GNU GPL license. Development continued at Sun and with a community of outside contributors, mostly sponsored by companies with an interest in helping their customers also use Java.

In 2010, Oracle bought Sun, including the few remaining proprietary bits of Java. As a result, multiple implementations of the Java virtual machine (JVM) have appeared. There is an implementation from Oracle, unsurprisingly called Oracle Java, which has some proprietary enhancements. The problem is that any code written using these enhancements will only run on the Oracle Java JVM that it was written for, violating the idea of “write once, run everywhere.” There is also a completely open source implementation called OpenJDK (Open Java Development Kit) that only includes code written or added using the same GNU GPL license. OpenJDK is sponsored by a community that includes Red Hat, IBM, Apple, and others. It is now considered the reference Java and is the one you should install and use unless an employer tells you otherwise because they use something else.

The Java language is class-based, object-oriented, and programs are compiled to bytecode that can run on any Java runtime on any platform (Linux, Windows, macOS, and others). Java uses an automatic garbage collector to manage memory, sparing programmers the burden of writing manual memory management. The syntax is similar to C++ (and was largely influenced by C++).

There are two Java packages in the Ubuntu repositories, both of which are OpenJDK related. One is the Java Runtime Environment (JRE), which is all you need to run Java programs; to use it, install the `default-jre` package. The other is the Java Development Kit (JDK), which is needed to write Java programs; to use it install the `default-jdk` package. If you install the JDK, the JRE features are included. The JDK also includes development and debugging tools and libraries.

There is also Kotlin, the new Java for Android development. See “Beginning Mobile Development for Android” later in this chapter for how to develop for Android. To just install the Kotlin compiler, install the Snap package with `sudo snap install kotlin --classic`.

Graphical Development Tools

This section branches out into information that more obviously applies to many languages. For example, Java is in widespread use, and you can develop in Java from Ubuntu along with any of the popular programming languages listed in Chapter 39, “Using Popular Programming Languages.”

Ubuntu has a number of graphical prototyping and development environments available. If you want to program in Java, for example, using your favorite integrated development environment (IDE) or a language with a standard software development kit (SDK), you can do that. If you want to build client software for KDE or GNOME, you might find the KDevelop, and Glade programs extremely helpful. You can use each of these programs to build graphical frameworks for interactive windowing clients, and you can use each of them to automatically generate the necessary skeleton of code needed to support a custom interface for your program.

IDEs and SDKs

IDEs and SDKs have become extremely popular. Although some programmers still prefer to write and edit software using a standard text editor, such as `nano` or `vi` (covered in Chapter 12, “Command-Line Master Class, Part 2”), many prefer using a tool that is more powerful. One commonly used tool, `emacs`, started out as a text editor, but as more and more features were added, it evolved into something more (see Chapter 12). By adding tools and features to make the programmer’s life easier, `emacs` unintentionally became the template for modern IDEs.

Some IDEs support multiple languages, like `emacs` does. Others focus on only one language. Most include not only programming language-specific features like code highlighting to help you read and browse code more quickly and efficiently, but also contain a compiler and debugger and even build automation tools. If you read through the details earlier in this chapter of using `make` with C, you can understand the value added.

So, what is the downside? Well, you can’t run a typical IDE on a server because you need a graphical interface, so if you are working on code that will run on a server that only has a command line or text interface available to you, you need to make sure you are comfortable with traditional methods. This doesn’t mean you can’t use a local desktop machine for development using an IDE and then push your code out to the server, but it means you should cover your bases—just in case.

The most commonly used IDEs seem to also be used most frequently by Java developers. We discuss several of them in this section. You should download these IDEs directly from the providers to ensure that you install the most current and standard versions.

Eclipse was originally created by IBM but has been spun off to a foundation created just for it. The nonprofit Eclipse Foundation coordinates efforts of volunteers and companies that contribute time, money, and code to this open source project. Eclipse is very widely used and popular. It supports multiple languages, and many plug-ins are available to extend its capabilities.

NetBeans is an extremely popular IDE that works with multiple languages. It is now owned by Oracle but was started by student programmers who were looking to create more useful tools for their needs. Others asked to contribute code, and soon NetBeans developed into a commercial program with plug-ins to extend its capabilities, many contributed by a large supporting community. Sun Microsystems, which developed and owned Java, bought NetBeans and released it under an open source license. When Oracle acquired Sun, it also acquired NetBeans; due to its popularity, it is worth a look. You can learn about and download NetBeans from <https://netbeans.org>.

Visual Studio Code is built on open source, which means it open source code along with proprietary code. The website provides a `.deb` download for installation on Ubuntu. It is pretty and comes highly recommended by developers who use it. You can learn more and download it from <https://code.visualstudio.com/>.

Oracle, which owns Java, provides an IDE for Java called **Oracle JDeveloper**. It is most commonly used in enterprise settings, where a team of developers work together using a standard tool. It is the least popular of the options mentioned here. You can learn more about it at www.oracle.com/technetwork/developer-tools/jdev/overview/index.html.

An SDK is a set of software development tools that are focused not on one language but on something narrower, such as one software package or framework (for example, the Android development SDK, described in the later section, “Beginning Mobile Development for Android”). A company may provide an SDK when it wants to encourage outsiders to write programs that run on the company’s product, such as its platform (like a game system from Nintendo or Sega) or operating system (like Android or iOS). Many open source enthusiasts will not participate in writing code for these platforms, so SDKs are less popular in this environment than they are on Windows and other platforms. Also, depending on the software license used to release the SDK, the potential uses of the code produced using the SDK can be limited, and not everyone is comfortable with those limitations. However, many SDKs are in use, and if you want to write code for a project that releases an SDK, it is likely to contain useful code examples, tools, and documentation to make the task much easier. Do your homework and make a choice that you are comfortable with.

Using the KDevelop Client

You can launch the KDevelop client from the applications menu or from the command line of a terminal window, like this:

```
matthew@seymour:~$ kdevelop &
```

After you press Enter, the KDevelop Setup Wizard runs, and you are taken through several short wizard dialogs that set up and ensure a stable build environment. You must then run `kdevelop` again (either from the command line or by clicking its menu item under the desktop panel's Programming menu). You then see the main KDevelop window and can start your project by selecting KDevelop's Project menu and clicking the New menu item.

You can begin building your project by stepping through the wizard dialogs. When you click the Create button, KDevelop automatically generates all the files that are normally found in a KDE client source directory (including the `configure` script, which checks dependencies and builds the client's makefile). To test your client, you can either first click the Build menu's Make menu item (or press F8) or just click the Execute menu item (or press F9), and the client is built automatically. You can use KDevelop to create KDE clients, plug-ins for the `Konqueror` browser, KDE `kicker` panel applets, KDE desktop themes, Qt library-based clients, and even programs for GNOME.

The Glade Client for Developing in GNOME

If you prefer to use GNOME and its development tools, the Glade GTK+ GUI builder can help you save time and effort when building a basic skeleton for a program. You launch Glade from the desktop panel's Programming menu.

When you launch Glade, a directory named `Projects` is created in your home directory, and you see a main window. You can use Glade's File menu to save the blank project and then start building your client by clicking and adding user interface elements from the Palette window. For example, you can first click the Palette window's Gnome button and then click to create your new client's main window. A window with a menu and a toolbar appears—the basic framework for a new GNOME client.

Beginning Mobile Development for Android

Many Linux users have embraced not only smart phones but specifically those based on Android. Android, owned by Google and based on the Linux kernel, is one of the best-selling platforms for smart phones and tablet computers. The Android platform includes the operating system, middleware, and several key applications. Middleware and application examples include an integrated web browser based on WebKit, optimized graphics libraries, media support for most formats, and structured data storage with SQLite. It also includes software for hardware-dependent functions such as GSM, Bluetooth, 3G, Wi-Fi, camera, GPS, and more.

Most of the Android source code is freely available and licensed using the Apache License. Google operates an online app store called *Google Play*, where users of Android phones or tablet computers can download free and for-payment applications to extend the functionality of their devices. Other third-party sites exist for the same purpose, thereby creating many paths for making software available to Android users.

This section helps you get started writing software for Android on your Ubuntu machine by describing how to find and set up the development tools you need. It discusses the basic setup details for developing Android software.

Before we get further into the details of developing software for Android, a more detailed introduction to the Android architecture is appropriate. Our description starts with the hardware and builds layer upon layer from that foundation.

Hardware

Although it has been proved possible to run Android on other platforms, the main target platform is ARM. ARM processors are 32-bit or 64-bit *reduced instruction set computer (RISC)* processors. Like other RISC processors, they are designed for speed, with the idea that a simpler set of processor instructions creates greater efficiency and throughput. ARM processors are also designed for low power usage, making them ideal for mobile and embedded devices. Indeed, ARM is the dominant processor in these markets.

Linux Kernel

The first layer of software to run in the Android stack is a customized Linux kernel. Most of the customizations take the form of feature enhancements or optimizations to help Android and Linux work together more efficiently. Originally, Google made a point of contributing code it developed, but some of the features were rejected by the mainline Linux kernel developers for inclusion in the standard Linux kernel. This meant that to keep its desired code customizations, Google had to create a fork of the Linux kernel, which is permissible due to the license under which the kernel is released. Chapter 22, “Kernel and Module Management,” provides an introduction to the Linux kernel.

Libraries

Software libraries run on top of the kernel. These libraries are used by the higher-level components of Android and are made available to developers to use when writing Android applications using the Android software development kit (SDK), which is discussed later in this chapter. These libraries include a version of the standard C library (`libc`), libraries for recording and playback of many popular media formats, graphics and web browser engines, font rendering, and more.

Android Runtime

Some of the higher-level components of Android in the Application layer (described next) interact directly with the libraries just described. Other parts of the Application layer interact with the libraries via the Android Runtime. Android software is primarily written in Java, using Google-developed and -specific Java libraries. That software runs on the Android Runtime, composed of some additional core libraries running on top of a special virtual machine called Dalvik. The core libraries provide most of the functionality of Java. Dalvik performs *just-in-time (JIT)* compilation and is optimized for mobile devices.

Application Framework

The Application Framework is a set of useful systems and services that top-level applications can call. These systems and services provide standardized means of accessing system information, using device hardware, creating notifications, and so on. They are the same

set used by the core applications included in Android, so end user-created applications can have the same look, feel, and interaction style as those provided by Android.

Applications

Android comes with a set of core applications, including a web browser, programs for text messaging, managing contacts, a calendar, an email client, and more. As noted earlier, Android software is written in Java.

Installing Android Studio

Android provides a bundled integrated development environment (IDE) with the software development kit, which is a set of tools to enable the creation of applications to run on Android. Android Studio has versions available for Linux, macOS, and Windows.

Download the latest version of Android Studio from the Android Developers website at <https://developer.android.com/studio/index.html>. For Ubuntu, you need the Linux version, which is made available as a `.zip` file. Unpack the file in the location where you want the development kit to reside (for example, `/home/matthew`). Doing so creates a new directory called `android-studio`. Note where you put this directory; you will need the information later.

Navigate to the `android-studio/bin/` directory and run `studio.sh`:

```
matthew@seymour:~$ studio.sh
```

The first time you run Android Studio, a wizard walks you through the initial setup procedure and then downloads and installs any basic components you need.

Creating Your First Android Application

After you have installed Android Studio and all the necessary SDK packages, you are ready to begin. Click Start a new Android Studio Project and use the wizard to enter the basic details of your new application.

Version Control Systems

Deciding whether to include information on version control systems in this chapter was difficult. On one hand, someone who only wants to scratch an itch quickly may not be interested in setting up a version control system. On the other hand, these systems are not difficult to set up, especially when used with the assistance of a code hosting site like the ones discussed in this chapter, and they are immensely valuable if code is to have a life outside your system.

Although you can use `make` to manage a small software project (see Chapter 38, “Using Programming Tools”), larger software projects require document management, source code controls, security, and revision tracking as the source code goes through a series of changes during its development. Version control systems provide utilities for this kind of large software project management. Changes to files placed in version control are tracked.

Files can be checked out by one developer, changed in their local environment, and tested before those changes are saved in the version control system. Changes that are later discovered to be unwanted can be found and removed from the tracked files. Various version control systems manage projects differently; some use a central repository, others a distributed format where any and every copy could become the master copy.

The next few sections introduce the most commonly used version control systems at the moment: Git, Bazaar, Subversion, and Mercurial. You have certainly heard of others, and new ones crop up every few years. Each has strengths and benefits. At the end of the chapter, in the “References” section, you can find a list of resources for learning more about these version control systems to further your knowledge after you peruse this chapter’s short and basic introduction to each one.

NOTE

Subversion and Mercurial are still in heavy use, but most developers today have switched to Git and Bazaar for new projects. Keep this in mind as you read the next few sections.

Managing Software Projects with Git

Git, initially created by Linux kernel creator Linus Torvalds, was first released in 2005 to host all development files for the Linux kernel. It is now actively developed by a large team of developers led by Junio Hamano and is widely used by many other open source projects.

Git works without a central repository, and it comes from a different perspective than other version control systems while accomplishing the same goals. Every directory that is tracked by Git acts as an individual repository with full history and source changes for whatever is contained in it. There is no need for central tracking. Source code control is done from the command line, as shown in the following examples. You need to install Git from the Ubuntu software repositories, where it is called `git`.

To create a new repository, access the top-level directory for the project and enter the following:

```
matthew@seymour:~$ git init
```

To check out code from an existing central repository, you must first tell Git where that repository is:

```
matthew@seymour:~$ git remote add origin git://path_to_repository/directory/proj.git
```

Then you can pull the code from that repository to your local one:

```
matthew@seymour:~$ git pull git://path_to_repository/directory/proj.git
```

To add new files to the repository, use the following:

```
matthew@seymour:~$ git add file_or_dir_name
```

To delete files from the repository, use this:

```
matthew@seymour:~$ git rm file_or_dir_name
```

To check in code after you have made changes, you will need to set your email and name in your `.gitconfig` file using `git config --global user.name` and `git config --global user.email` for this to work.

Then, use the `-m` flag to add a note, which is a good idea to help others understand what the commit contains:

```
matthew@seymour:~$ git commit -m 'This fixes bug 204982.'
```

In Git, a `commit` does not change the remote files but only commits the change to your local copy. If you want others to see your changes, you must `push` the changes to them:

```
matthew@seymour:~$ git push git://path_to_repository/directory/proj.git
```

Many open source projects that use Git host their code using GitHub. You can find it at <https://github.com>.

Managing Software Projects with Bazaar

Bazaar was created by Canonical and first released in 2007 to host all development files for Ubuntu and other projects. It is actively developed and used by Canonical and Ubuntu developers and also by other open source projects. Launchpad, covered later in this chapter, uses Bazaar.

Bazaar supports working with or without a central repository. Changes are tracked over any and all files you check out, including multiple versions of files. Source code control is done from the command line, as shown in the following examples. You need to install Bazaar from the Ubuntu software repositories, where it is called `bzr`.

There are two ways to create a new repository. If you are starting with an empty directory, use the following:

```
matthew@seymour:~$ bzr init your_project_name
```

If you are creating a repository for an existing project, go to the top-level directory for the project and enter the following:

```
matthew@seymour:~$ bzr init
matthew@seymour:~$ bzr add .
```

To check out code from an existing central repository, use this:

```
matthew@seymour:~$ bzr checkout your_project_name
```

To check your changes before you check them in, you can use `bzr diff` or `bzr cdiff`. They do the same thing, but `bzr cdiff` does so with colored output:

```
matthew@seymour:~$ bzr cdiff
```

To check in code after you have made changes, use the `-m` flag to add a note, which is a good idea so that others know what the `commit` contains:

```
matthew@seymour:~$ bzip commit -m "This fixes bug 204982."
```

In Bazaar, a `commit` does not change the remote files but only commits the change to your local copy. If you want others to see your changes, you must `push` the changes to them:

```
matthew@seymour:~$ bzip push sftp://path.to.main/repository
```

To update the source code in your local repository from the main repository to make sure you have all the latest changes to the code from other developers, use the following:

```
matthew@seymour:~$ bzip pull
```

Many open source projects that use Bazaar host their code using Launchpad, which is where Ubuntu development takes place. You can find more about it later in this chapter and also at <https://launchpad.net>.

Managing Software Projects with Subversion

Subversion was first created in 2000 as a replacement for an older version control system called the *Concurrent Versions System (CVS)*. At that time, CVS was 10 years old, and although it served its purpose well, it lacked some features that developers wanted. Subversion is now actively developed and widely used.

In Subversion, you check out a file from a repository where code is stored in a client/server fashion. Then, changes are tracked over any and all files you check out, including multiple versions of files. You can use Subversion to backtrack or branch off versions of documents inside the scope of a project. It can also be used to prevent or resolve conflicting entries or changes made to source code files by multiple developers. Source code control with Subversion is done from the command line, as shown in the following examples. You first need to install Subversion from the Ubuntu software repositories, where it is called `subversion`.

You can create a new repository as follows:

```
matthew@seymour:~$ svnadmin create /path/to/your_svn_repo_name
```

To add a new project to the repository, go to the top directory of the code that is going to be placed into the repository. Then create three subdirectories: `branches`, `tags`, and `trunk`. Move all of your files into `trunk` and enter the following:

```
matthew@seymour:~$ svn import project file:///your_svn_repo_name/your_project -m
"First Import"
```

To check out code from an existing central repository, use this:

```
matthew@seymour:~$ svn checkout file:///your_svn_repo_name/your_project/trunk
your_project
```

To check in code after you have made changes, use the `-m` flag to add a note, which is a good idea so that others know what the `commit` contains:

```
matthew@seymour:~$ svn commit -m "This fixes bug 204982."
```

To update the source code in your local repository from the main repository to make sure you have all the latest changes to the code from other developers, use this:

```
matthew@seymour:~$ svn update
```

To add new files to the repository, use the following:

```
matthew@seymour:~$ svn add file_or_dir_name
```

To delete files from the repository, use this:

```
matthew@seymour:~$ svn delete file_or_dir_name
```

Many open source projects that use Subversion host their code using SourceForge, which also works with Git. You can find it at <https://sourceforge.net>.

Continuous Integration and Continuous Delivery and DevOps Tools

Continuous integration and continuous delivery (CI/CD) is the combined practice of first merging all the various developers' work into a shared main branch several times a day and then building, testing, and releasing that software with greater velocity. The practice has become an integral part of DevOps. The implementation is called a *CI/CD pipeline*. Let's start with the high-level view.

Continuous integration (CI) involves a set of practices that your team agrees to use. When a team member is working on a feature or a bug fix, she checks out the most current *master* branch of the code from the code repository. The team member writes her new code and after checking that it works locally immediately checks the changes in to that master branch. Changes are agreed to be as small as possible (often called *atomic changes*). Small changes are easy to roll back if problems occur, and it is easy to track down what is causing the problem if only a small amount of code needs to be reviewed.

The goal of CI is to first establish a consistent way for teams to work, and then to automate the process of building, packaging, and testing applications. As this automation is created, consistency in the integration process is achieved, which makes it even easier for team members to check out and in small changes frequently. This stage typically implements a form of continuous testing into the CI.

The largest benefit comes when you add continuous delivery (CD), which automates the delivery of applications that pass the integration process. This could be delivery to a testing or staging environment or even to a production environment (this last option changes "continuous delivery" to "continuous deployment").

CI/CD improves team collaboration, code quality, and pushes changes out to production much more quickly. Whether the changes are bug fixes or new/improved features, a greater velocity of change benefits and pleases customers.

CI/CD Tools

Many tool types are useful in a DevOps CI/CD context. Each has its place and some of the tools available fit into multiple categories. It is up to your team (or leadership) to select the toolchain that is right for your context.

The main type of tool is the automation server, which manages the CI/CD process. Popular options include Jenkins, Maven, Travis CI, and Spinnaker.

Another type of tool is for configuration management, which is important, especially in a cloud-computing context. The use of containers will inform the toolchain options your team selects. See Chapters 31, 32, and 33 for more on these technologies.

Chaos Engineering

Started by Netflix as a way to find out whether its servers could withstand production problems in its cloud host's infrastructure, Chaos Engineering aims to test large-scale cloud deployments in ways no other testing can. The goal is to find systemic problems and their causes before there are customer-impacting failures. This is done by intentionally and carefully simulating failures in things like networking, DNS, or system resource use to see how those and other issues impact the system as a whole. When problems are found, mitigation schemes and automated failover methods can be implemented to enhance overall system reliability and resilience.

Most DevOps and Site Reliability Engineering teams are implementing Chaos Engineering into every part of their process, from early in the CI/CD pipeline through any testing or staging environments and even into production. Why production? Because today's large-scale cloud applications are constantly changing and no testing or staging environment can accurately mimic what an application service or microservice will experience in production.

For transparency, the book's author works for a company, Gremlin (<https://gremlin.com>), that provides a Software as a Service Chaos Engineering implementation. Many open source options are also available for teams to implement themselves.

Canonical-created Tools

The tools in this section are created and used by Canonical for Ubuntu development. They are not in widespread use elsewhere.

Launchpad

To get started with Launchpad, you need to sign up for an account on Launchpad. Launchpad is an infrastructure created to simplify communication, collaboration, and processes within software development.

Launchpad is where much of Ubuntu development takes place, although some has moved to Git. It integrates Bazaar, the version control system introduced earlier, to make keeping track of changes to software code much simpler and to permit those changes to be reverted when necessary while tracking who is performing the actions.

For developers using Launchpad, this means that the process has become a bit simpler. They can concentrate on writing and editing their code and let Launchpad deal with keeping track of the changes and creating their packages. This is useful for active developers who write and maintain big projects that need source code version control and so on. Launchpad also hosts bug reporting and tracking, mailing lists, software interface translation, and much more.

Launchpad users can create a *personal package archive (PPA)*. This is a much simpler way to make programs available. Anyone with a PPA can upload source code to be built in to packages. Those packages will then be made available in an `apt` repository that can be added to any Ubuntu user's list of source repositories and downloaded or removed using any of the standard package management tools in Ubuntu, such as `apt`, Ubuntu Software Center, and Synaptic. Instructions are included on the web page for each Launchpad PPA, describing how to add that repository, which makes this an easy way to share software that may be added and removed by even nontechnical end users.

Ubuntu Make

Ubuntu Make is a command-line tool that sets up your system for development use quickly and easily. It allows you to download the latest version of popular developer tools and their dependencies, enable multi-arch, and more. Install `ubuntu-make` to get started. Then run commands like this to install tools:

```
matthew@seymour:~$ umake android
```

After you enter this, you are prompted to accept the install path and Google license, and then Ubuntu Make downloads and installs Android Studio and the latest SDK, configures everything, and, if you are running on a 64-bit platform, it even adds a Unity launcher icon.

See <https://wiki.ubuntu.com/ubuntu-make> to learn more about the packages available and get started.

Creating Snap Packages

Snap packaging allows a single packaged application to be used across multiple Linux distributions. Although snap packages are not expected to replace traditional packaging formats like `.deb`, which we cover in Chapter 40, “Helping with Ubuntu Development,” it is reasonable to expect snaps to find wide use for applications provided by third-party vendors—for example, Mozilla is already committed to using snap packages for its Firefox web browser—and for applications intended for use on devices such as phones, routers, switches, and the new category of IoT (Internet of Things) devices (see https://en.wikipedia.org/wiki/Internet_of_things). For desktop applications, snap packaging enables a developer to submit free or even for-payment apps for review and inclusion in the Ubuntu Software application (see Chapter 9, “Managing Software”).

The tool used to create snap packages is Snapcraft, available from <https://snapcraft.io>. Snapcraft is designed to bundle your already-created application with any and all dependencies for easy installation and updating.

Another helpful community project related to snap packaging is Snappy Playpen, at <https://github.com/ubuntu/snappy-playpen>, which exists to share knowledge and best practices about snap packaging while helping test the packages that community members and others create.

Bikeshed and Other Tools

Bikeshed was started by Dustin Kirkland in September 2010 as a project to package a series of tools he wrote to scratch some personal itches that he had as an Ubuntu developer working on Canonical's Ubuntu server team or that he thought would be useful to others. All good developers, system administrators, and DevOps gurus eventually write scripts to perform specific tasks they find useful. The Bikeshed project began when Dustin gathered his scripts together and made them accessible to the world. The wider Ubuntu community is invited to give suggestions or submit patches to make them better.

Bikeshed sometimes works as an incubator, housing specific tools until they are ready to stand alone as a separate package or until they are accepted into an existing package. All the tools run from the command line, and most have useful man pages. (Others are still being written.) The project describes itself as “a collection of random useful tools and utilities that either do not quite fit anywhere else, or have not yet been accepted by a more appropriate project. Think of this package as an ‘orphanage,’ where tools live until they are adopted by loving, accepting parents.” The slogan for Bikeshed on the Launchpad project page is “While others debate where some tool should go, we put it in the Bikeshed.”

NOTE

To give credit where credit is due, much of the content in this section comes, with permission, from Dustin's blog, at <https://blog.dustinkirkland.com>, from direct communication with him, and from the tool man pages. Dustin also wrote Byobu, a tool that is covered at the end of Chapter 12 and that contains some of the tools that have graduated from Bikeshed.

You can get the following tools by installing the Bikeshed package from the Ubuntu repositories:

- ▶ **apply-patch**—Wraps the `patch` utility and makes it a little easier to use by automatically detecting the `patch` strip level.
- ▶ **bch**—Determines what files have been modified in the current Bazaar (`bzr`) tree, opens `debian/changelog` for editing, uses `dch`, and appends a `changelog` entry for the current list of modified files.
- ▶ **bzrp**—Operates the same as `bzr` except that output is piped to a pager to make reading easier.

- ▶ **cloud-sandbox**—Launches a cloud instance and connects directly to it by using `ssh`, with the cloud system running isolated, as what is generally called a *sandbox*.
- ▶ **dman**—Remotely retrieves man pages from <https://manpages.ubuntu.com> but reads them on the local system. This is useful for reading the man page for a utility you do not have installed on the local system.
- ▶ **pbget**—Retrieves content uploaded to a pastebin by `pbput` or `pbputs`.
- ▶ **pbput**—Uploads text files, binary files, or entire directory structures to a pastebin. It is similar to `pastebinit`, described later, but adds support for binaries and only uses <https://pastebin.com>.
- ▶ **pbputs**—Operates exactly like `pbput`, except the user is prompted for a passphrase for encrypting the content with `gpg` before uploading. `pbget` automatically prompts the user for the preshared passphrase when the file is requested.
- ▶ **release**—Creates a release of a project for Ubuntu.
- ▶ **release-build**—Takes project information for a `bzr` project in a Launchpad PPA that uses specific parameters and builds the project as an upstream project that can then be released to Ubuntu.
- ▶ **release**—Creates a release of a project for Ubuntu.
- ▶ **socks-prox**—Establishes an encrypted connection for tunneling traffic through a `socks proxy`.
- ▶ **system-search**—Performs a unified search through a set of system commands, packages, documentation, and files.
- ▶ **uquick**—Performs a quick server installation.
- ▶ **what-provides**—Determines which package provides a specific binary in your path.

The contents of Bikeshed are expected to change over time. Some of these tools may graduate to standalone tools, merge into other existing packages, or get added to more official upstream packages. You can always check the Launchpad page to find a current list of Bikeshed's contents.

The rest of the tools in this section are not actually part of Bikeshed but have either graduated from Bikeshed and been spun off as freestanding tools or were developed individually by Dustin or others in the Ubuntu community. All the tools run from the command line and have useful man pages.

Other useful tools that you can find in the Ubuntu repositories include the following:

- ▶ **pastebinit**—Uploads a file or the result of a command to the pastebin you want and gives you the URL in return. It was written by Ubuntu developer Stéphane Graber, and you can find it at <https://launchpad.net/pastebinit> or from the Ubuntu repositories. By default, it uses <https://pastebin.com>, but it can be configured to use others, such as <https://paste.ubuntu.com>.

- ▶ **run-one**—Runs no more than one unique instance of a command with a unique set of arguments. This is often useful with `cron` jobs, when you want no more than one copy running at a time but where a `cron` job has the potential to run long and finish after the next scheduled run. Also see `run-one-constantly`, `run-one-until-failure`, and `run-one-until-success` in the man page.
- ▶ **run-this-one**—Operates exactly like `run-one` except that it uses `pgrep` and `kill` to find and kill any running processes owned by the user and matching the target commands and arguments. It blocks while trying to kill matching processes until all matching processes are dead.
- ▶ **keep-one-running**—Operates exactly like `run-one` except that it respawns "`COMMAND [ARGS]`" any time `COMMAND` exits (zero or nonzero).
- ▶ **ssh-import-id**—Uses a secure connection to contact a public key server (`https://launchpad.net` by default) to retrieve one or more users' public keys and append them to the current user's `~/.ssh/authorized_keys` file.
- ▶ **bootmail**—Called by `cron` to send an email any time a system is rebooted. It reads a list of one or more comma-separated email addresses from `/etc/bootmail/recipients` and then loops over a list of white space-separated files in `/etc/bootmail/logs` to construct the email. This is useful for knowing when remote systems are rebooted.
- ▶ **purge-old-kernels**—Looks for old kernels on your system and removes them. This is a part of the `Byobu` package.
- ▶ **col1**—Splits and prints a given column, where the column to print is the name of the script program you are running (`col1` to `col9`). `col2` to `col9` are symlinks to `col1`; their behavior simply changes based on the name called. For example, instead of using `awk '{print $5}'`, you can use `col5`. This used to be in `Bikeshed` but is now part of the `Byobu` package.
- ▶ **wifi-status**—Monitors a wireless interface for connection and associated information. This used to be in `Bikeshed` but is now part of the `Byobu` package.

References

- ▶ **www.cprogramming.com**—A useful website for learning C and C++
- ▶ **https://gcc.gnu.org**—The main website for `gcc`, the GNU Compiler Collection
- ▶ **www.gnu.org/software/autoconf/autoconf.html**—More information about the GNU Project's `autoconf` utility and how to build portable software projects
- ▶ **www.qt.io**—The main Qt website
- ▶ **https://glade.gnome.org**—Home page for the Glade GNOME developer's tool
- ▶ **www.kdevelop.org**—Site that hosts the KDevelop Project's latest versions of the KDE graphical development environment, KDevelop

- ▶ ***The C Programming Language* by Brian W. Kernighan and Dennis M. Ritchie**—The standard resource for learning C
- ▶ ***The Annotated C++ Reference Manual* by Margaret A. Ellis and Bjarne Stroustrup**—An excellent resource for C++
- ▶ **<https://subversion.apache.org>**—The main website for Subversion
- ▶ **<https://bazaar.canonical.com/en/>**—The main website for Bazaar
- ▶ **<https://git-scm.com>**—The main website for Git
- ▶ **<https://jenkins.io>**—The main website for Jenkins
- ▶ **<https://maven.apache.org>**—The main website for Maven
- ▶ **<https://travis-ci.org>**—The main website for Travis CI
- ▶ **<https://spinnaker.io>**—The main website for Spinnaker
- ▶ **<https://launchpad.net>**—An open source website that hosts code, tracks bugs, and helps developers and users collaborate and share
- ▶ **<https://launchpad.net/ubuntu/+ppas>**—Personal package archives that allow source code to be uploaded and built into `.deb` packages and made available to others as an `apt` repository
- ▶ **<https://glade.gnome.org>**—A user interface designer for GTK
- ▶ **<https://launchpad.net/bikeshed>**—The main page for Bikeshed
- ▶ **<https://developer.android.com/>**—The main website for Android development. Most of this chapter could not exist if it were not for this site, which goes into much greater detail than this short introduction and is where we learned most of what we know on the subject.
- ▶ **<https://developer.android.com/sdk/>**—This is the main web page for Android Studio.

This page intentionally left blank

CHAPTER 39

Using Popular Programming Languages

Students and developers who pay attention to either computer programming history or current trends are already aware of a vast array of possibilities that deserve some attention. The goal of this chapter is to introduce many different languages available for you to use, some that are old and some that are new.

This short chapter does not teach you how to use these languages but rather exposes you to them, making you aware of their existence and giving just enough information about them to help you decide whether each one sounds interesting or useful to you; this chapter also points to resources to help guide your next steps. The introductions include information for installing and getting started with each language on Ubuntu. Sometimes the version of the language included in the Ubuntu repositories is a little outdated or is a free (as in freedom) version instead of an official, proprietary version. If you discover you need a more up-to-the-minute version or a one that is not mentioned, check the “References” section to find links to more information for most languages.

The organization of this chapter was tricky. The original idea was to order the chapter by how well known a language is, but that is problematic. Older languages generally have better name recognition, even though newer ones might be more commonly used. How do we measure popularity and use it to enforce order on a list of programming languages? The people who use them tend to form strong emotional bonds with those they prefer, just as geeks do with text editors (`emacs` versus `vi`), email programs (web based versus Thunderbird versus Mutt), and more important matters like comic book universes (Marvel versus DC). Ultimately, it seemed like alphabetic order would be best. If your favorite language is not included in this edition, and

IN THIS CHAPTER

- ▶ Ada
- ▶ Clojure
- ▶ COBOL
- ▶ D
- ▶ Dart
- ▶ Elixir
- ▶ Elm
- ▶ Erlang
- ▶ Forth
- ▶ Fortran
- ▶ Go
- ▶ Groovy
- ▶ Haskell
- ▶ Java
- ▶ JavaScript
- ▶ Kotlin
- ▶ Lisp
- ▶ Lua
- ▶ Mono
- ▶ OCaml
- ▶ Perl
- ▶ PHP
- ▶ Python
- ▶ Raku
- ▶ Ruby
- ▶ Rust
- ▶ Scala
- ▶ Scratch
- ▶ Vala
- ▶ References

you think it should be included in future editions, please email your suggestion and reasoning to the author at matthew@matthewhelmke.com for consideration.

You will notice an interesting mix of old and new languages here. Some have asked why some of the languages are included, especially the older ones. They are included because they are still in use in the real world. There may not be many or even any new projects being created using some of these languages, but if you are attentive, you are likely to find each of them in use somewhere. A 2013 quote from *The Register* illustrates this beautifully: “The venerable PDP-11 minicomputer is still spry to this day, powering GE nuclear power-plant robots—and will do so for another 37 years. That’s right: PDP-11 assembler coders are hard to find, but the nuclear industry is planning on keeping the 16-bit machines ticking over until 2050—long enough for a couple of generations of programmers to come and go” (from www.theregister.co.uk/2013/06/19/nuke_plants_to_keep_pdp11_until_2050/). The PDP-11 came out in 1970 and has not been produced since the early 1990s. Programming languages, like programs, always seem to outlast their expected or intended life span. Who knows? Learning that old language that everyone laughs about may benefit you with a well-paying, unique job as well as help you to think about human–computer communication in a different way.

Ada

Ada is based on Pascal. It is named after Ada Lovelace (1815–1852), who wrote the first algorithm designed to be processed by a machine—specifically the mechanical computing device created by Charles Babbage.

The language is most known for its use in embedded systems, especially in the aeronautics and avionics realm. Ada is well known as a reliable and efficient real-time language. It is most commonly encountered in aircraft systems, air traffic control and railroad systems, and medical devices. It is also often used as a teaching language for computer science courses.

Language highlights include static typing, concurrency, synchronous message passing, protected objects, modularization, and exception handling. Ada is object oriented, has standard libraries for things like I/O and containers, has good interfaces to other languages like C, and works well with distributed systems and numerical processing.

To use Ada on Ubuntu, you write programs in your favorite text editor. To compile, you need the package `gnat`, which is the *GNU Ada Compiler*. You may want to consider `gnat-gps`, which installs the Gnat Programming System, an *integrated development environment (IDE)* specifically for Ada and C programming.

Clojure

Clojure is a newer dialect of Lisp that runs on the *Java Virtual Machine (JVM)*. It is intended for general-purpose use. It encourages functional programming and is designed to make writing multithreaded applications easier. Because of the close integration with Java, Clojure applications can be packaged and deployed to JVM environments without adding complexity. It also provides easy access to Java frameworks, and Clojure’s data structures all implement standard Java interfaces.

Closures are a common element in programming, especially in functional programming languages, such as Clojure. (The language's name is rumored to be a mash-up of *closure* and *Java*.) Typically, variables are designed to work only within a defined function. Using a closure is a way to bend that rule temporarily. A closure starts with a function and allows the value of one or more variables from that function to be available outside that function while being maintained in the function. Another way to phrase that is to say a closure is a combination of a function and the variables that were in scope at the time the function was defined; the function can refer to those variables even if they are no longer in scope when the function is called. When you are working in a programming language with first-class functions that can be passed around like variables, using closures is a convenient way to provide encapsulation without using objects or classes. An example of the use of a closure is a function being encapsulated completely within another function but still being able to read the state of a variable that exists in the containing function.

Clojure is unlike most other languages in that you don't generally install Clojure itself; it's just a library that's loaded into the JVM. You don't interact with it directly but use a build tool and editor/IDE integration instead (well, except when you are interacting using the REPL; in any case, you do not typically run a tool called `clojure` from the command line other than the REPL). That process is a bit beyond the short introduction of this chapter; however, you can get started quickly by installing the `clojure` package. Among other things, installing this package also installs a REPL (read-eval-print loop, an interactive programming environment) that can be used by entering `clojure` at the command line. When you install this package, you get what you need for using Clojure in a JVM, but you need to set up your development environment to use Clojure. See the documentation from each environment for instructions. Note that the most popular, perhaps even the de facto, build tool for Clojure is Leiningen, which adds some really useful functionality, such as the ability to manage project dependencies, start a REPL easily, and even install Clojure itself. See <https://github.com/technomancy/leiningen#readme> for more.

COBOL

COBOL, *Common Business Oriented Language*, has been around since the 1960s. It is one of the oldest programming languages and is still the dominant language for (legacy) business applications. The majority of big business applications, such as payroll and accounting, were written in COBOL, and most programmers who knew it well have retired. A job market exists for younger programmers willing to learn it and willing to support older applications that are stable and trusted and still running all over the place. This is especially true if you are able to understand the business processes modeled in COBOL and can integrate it with modern technology, which isn't always an easy task.

COBOL's syntax was designed to mimic natural human language. Often a newcomer can read COBOL source code and have a pretty good idea of what it does, even if the reader has little to no programming experience. An interesting feature in early COBOL that is deprecated in more recent versions is self-modifying code, which had the potential to create some interesting situations. COBOL has always been a little controversial, as illustrated by the time that Edsger Dijkstra remarked that "the use of COBOL cripples the mind; its teaching should, therefore, be regarded as a criminal offense."

In a 2012 survey in *ComputerWorld*, 46 percent of the respondents said they were already noticing a COBOL programmer shortage, while 50 percent said the average age of their COBOL staff was 45 or older and 22 percent said the age was 55 or older (see www.computerworld.com/s/article/9225099/Cobol_brain_drain_Survey_results). Organizations are trying to move away from COBOL, but it is still in extensive use.

To use COBOL on Ubuntu, you write programs in your favorite text editor. To compile, you need the package `open-cobol`, which actually translates the programs into C and compiles them using `gcc`.

D

The D language is a lot like C but much newer. It has similar syntax. It also has static typing. However, it has some differences that are designed for convenience, power, and continued efficiency. In D, you are able to write large amounts of code without redundantly specifying types. Static inference deduces types and other code properties. Also, memory management is automatic, and you have built-in linear and associative arrays, slices, and ranges. D has new methods of dealing with concurrency, scaling, and internal integration of features such that the presence of one feature does not harm another—for example, offering classic polymorphism, value semantics, functional style, contract programming, and more. Like C, D is compiled to native code.

We haven't really seen D in use in the real world yet, but it is receiving a lot of attention in the academic and research world. The buzz among programmers is that D is very promising and could become a successor to C, as it is intended to be.

To use D on Ubuntu, you write programs in your favorite text editor. To compile, you must first download and install a package from the D Programming Language website at <https://dlang.org/download.html>, following the instructions at the site.

Dart

Dart is a new language and an open source project headed up by Google. It is a suite of tools and libraries focused on scalable web application engineering. You write code in Dart, and it is compiled to JavaScript, which means that what you write will already be able to run in every major web browser and on nearly all servers. Dart is class based and object oriented. The code is concise without being enigmatic. The syntax looks very familiar and is pretty easy to figure out if you have some experience with other major languages. Dart allows you to create and use types, but it does not require types, and it is designed to be modular and scalable, offering the ability to organize your code with functions, classes, libraries, and so on. The compiler can weed extraneous libraries from your code, such as those you included but never used, during compilation to create a smaller application. It can even minify as it compiles to JavaScript.

To use Dart on Ubuntu, download it along with the editor and tools from the Dart website at www.dartlang.org.

Elixir

Elixir is a dynamic, functional language based on Erlang. It has an interactive mode and an executable mode. The main differences between Elixir and Erlang lie in the realm of convenience. Elixir has its own package management system, macros, and build tool. It is compatible with existing Erlang libraries. Its main use at the moment is for building scalable web-based applications.

To use Elixir on Ubuntu, install the package `elixir`.

Elm

Elm is a functional language that compiles to JavaScript. It is designed to help make websites and web apps with an emphasis on simplicity. The goal of simplicity is confidence while programming, confidence that nothing has been forgotten or misunderstood. Elm includes reliable refactoring, automatically enforced semantic versioning for all Elm packages, and friendly error messages.

To use Elm on Ubuntu, you have to download the official binary from a GitHub repo (official installers exist for Windows and macOS). The lack of official packaging for Linux may eliminate the possibility of using Elm in many organizations and companies, but if you are interested, take a look at the current install instructions at <https://guide.elm-lang.org/install/elm.html>.

Erlang

From the official website, “Erlang is a programming language used to build massively scalable soft real-time systems with requirements on high availability. Some of its uses are in telecoms, banking, e-commerce, computer telephony and instant messaging. Erlang’s runtime system has built-in support for concurrency, distribution and fault tolerance.” Erlang is a declarative, functional language that includes real-time garbage collection and hot-swapping of code. It is primarily designed for distributed applications that require extreme uptime.

Erlang’s greatest strength is its ability to create a large number of concurrent processes, each with low overhead, and allow them to communicate with one another using an asynchronous message handling system. In addition, the Erlang developers have a philosophy of development that emphasizes keeping things running, meaning that future stable Erlang updates should not break running code. They like to test extensively and try to break as much as possible in testing to prevent breakage in production systems. One of the technical editors mentioned a neat feature while reading this chapter: Erlang’s processes can communicate with each other even if they don’t live on the same box. You have to explicitly allow this in your code, but the ability is built in to the language runtime and syntax and is not a third-party library.

To use Erlang on Ubuntu, you write programs in your favorite text editor. To compile, you need the package `erlang`, which installs the Erlang/OTP runtime, applications, sources, code examples, and the Erlang editing mode for `emacs`.

Forth

Forth, which first appeared in the 1970s, is an interactive, procedural, imperative language with typeless data that runs as a shell. Sets of instructions can be saved and compiled as bytecode programs. It is a very small language by itself and is therefore very useful in boot loaders and embedded systems, and it has even been used by NASA in space applications.

On the surface, Forth is a simple language, but it is highly extensible. In essence, a programmer creates a dictionary, beginning with a small set of predefined words. These are combined in new ways to extend the lexicon and create new things that may be done. This is powerful, but it is also dangerous. The lack of standards can lead to less-than-stellar programmers creating unclear sets of vocabulary that make it impossible for anyone else to maintain what they have written. However, thoughtful programmers who are disciplined and organized have created highly complex, yet maintainable, programs in Forth that have been used for decades across multiple platforms.

Perhaps Forth might be described as a language for experienced programmers who can handle total control over the CPU and want to build sophisticated systems running in extremely limited environments. If you are old enough to remember HP calculators and their “reverse Polish” notation, where the operator is placed after the operands, you will find Forth familiar.

To use Forth on Ubuntu, install the package `gforth`, which is the GNU implementation of a Forth programming environment.

Fortran

Fortran was developed by IBM in the 1950s for engineering and scientific applications. Its popularity spread quickly in areas of science that are dominated by numerical computation. Today, many of those same Fortran programs are still maintained and in use in fields such as weather modeling and prediction, fluid dynamics, and segments of chemistry and physics. In a published article from 2010, Eugene Loh, an engineer at Oracle, called Fortran the most commonly used and perhaps the ideal language for high-performance computing (<https://queue.acm.org/detail.cfm?id=1820518>).

Fortran is a terse language in which complex applications may be written with relatively few statements. It is a procedural language with object-oriented abilities. It excels at numerical computation and is often used as the language in which programs are written to test supercomputers for speed.

To use Fortran on Ubuntu, you write programs in your favorite text editor. To compile, you need the package `gfortran`, the GNU Fortran 95 compiler, which compiles Fortran 95 on platforms supported by the `gcc` compiler. It uses the `gcc` back end to generate optimized code.

Go

Go is an open source project and language being developed by people at Google. It is an expressive language that aims for concise, clean code and efficient use of resources. It has

concurrency mechanisms built in to take advantage of multiple core machines and networked machines. Go includes a unique type system designed for flexible and modular code, and it's a compiled language that also has garbage collection. It is advertised as "a fast, statically typed, compiled language that feels like a dynamically typed, interpreted language."

To use Go on Ubuntu, write your programs in your favorite text editor and install the `golang` compiler. Install the `golang-docs` package for technical documentation.

Groovy

Groovy, like Clojure, is designed for the JVM. It was written to enable features like closures and dynamic typing from popular languages like Python and Ruby to be used by Java developers. It uses a Java-like syntax, making it familiar to those programmers. It can be compiled into standard Java bytecode and used within any Java project. It can also be used dynamically for scripting, templating, or writing unit tests.

To use Groovy on Ubuntu, you must first install a JVM. Then, you need the package `groovy`. You can then run Groovy code in a shell by entering `groovysh` at the command line or in an interactive console by entering `groovyConsole`, or you can run a specific Groovy script by entering the script file's name at the command line prefaced by `groovy`, like this:

```
matthew@seymour:~$ groovy scriptname.groovy
```

Haskell

Haskell is a purely functional programming language. It has built-in concurrency and parallelism and good support for integration with other languages. In that sense, it is similar to Erlang. From the beginning in 1990, it has been developed as an open-source project with strong community input and participation. Haskell uses lazy evaluation, meaning that the evaluation of an expression is put off until the last possible moment, until its value is required. This significantly speeds up runtime by avoiding unnecessary or repeated evaluation.

To use Haskell on Ubuntu, install `haskell-platform`, a suite of tools and libraries that contain the most important and best-supported components. It is meant to be a starting point for Haskell developers who are looking for libraries to use. To compile, you need the package `ghc`, which is the *Glorious Glasgow Haskell Compilation system (GHC)*, a compiler for Haskell.

Java

Java was created by Sun Microsystems, now owned by Oracle, as a write-once, run-anywhere language. Java programs are compiled to a bytecode that will run on any JVM. A JVM must exist on a hardware platform for Java code to run, but no recompilation of the program itself is needed for it to run on different hardware platforms. Java is object oriented, and writing program instructions for a virtual machine is generally easier than doing so for a real machine. Java syntax is similar to that of C and C++ but is a bit simpler

and has fewer low-level abilities. It is currently one of the most popular programming languages. Originally, Java technology was proprietary and licensed for use by Sun. In May 2007, several years before being bought by Oracle, Sun finished relicensing and releasing most Java technology under the GNU GPL. (There were parts Sun could not relicense because it did not own the copyright to the code.)

Java uses automatic garbage collection to remove objects from memory when they are no longer in use. This frees programmers from thinking about memory management. It includes a graphical user interface library called Swing.

Most Java development occurs in an IDE, several of which are available from the Ubuntu repositories. The most popular IDEs are Eclipse (www.eclipse.org) and NetBeans (www.netbeans.org). Each of these includes plug-ins that help a programmer include and use libraries, compile to bytecode, and do many other tasks quickly and efficiently.

To program in Java on Ubuntu, you write programs in your favorite text editor or IDE. To compile, you need the package `default-jdk`, which installs the Java Development Kit appropriate for the hardware being used.

JavaScript

JavaScript is an object-oriented, functional programming language designed primarily for scripting. It supports closures and dynamic and weak typing, and it has a syntax that is influenced by C and Java, even though it is unrelated to either (except for the circumstantial name similarity with Java). JavaScript was designed to be used by the Netscape web browser as a way to run short programs on web clients. The name is a result of a mid-1990s marketing agreement between Netscape and Sun to try to leverage the buzz about Java and make JavaScript the shiny, new programming language for the Web. You will occasionally see JavaScript referred to using its original name, EMCAScript. The JavaScript trademark is now owned by Oracle under a license from the technology creators, including Mozilla, the descendant of Netscape.

JavaScript is easily the most popular scripting language for the Web, widely used in programming web applications. Combined with HTML and CSS, it is used to create interesting, diverse, and powerful websites. JavaScript has spawned tons of extensions and development kits, such as Node.js and JSP. It is commonly combined with other technologies, like XML, to create interactive websites using Ajax. Information is often passed using *JavaScript Object Notation*, or *JSON*, which is rapidly becoming the successor to XML. Whether people love JavaScript or hate it, it is universally acknowledged as a “must-know” technology for programmers today.

To use JavaScript on Ubuntu, you write programs in your favorite text editor. Nothing special is needed. Put the script somewhere and open it with your web browser.

Kotlin

Kotlin is a statically typed, cross-platform language that runs on the Java virtual machine and can also be compiled to JavaScript source code, or you can use the LLVM compiler. It has a concise syntax and supports both object-oriented and functional programming

styles. It is open source and has seen use in mobile programming for Android, where it is becoming quite popular for new projects and is even supported in the Android Studio 3.0 IDE as an alternative to the standard Java compiler. Kotlin was created and is supported by JetBrains, which is a global software vendor.

To use Kotlin on Ubuntu, install the snap package using

```
matthew@seymour:~$ snap install --classic kotlin
```

Lisp

Lisp is slightly younger than Fortran—first released in 1958—making it not quite the oldest language discussed in this chapter. Clojure, discussed earlier, is a dialect of Lisp. Lisp is designed to process lists. Linked lists are the language’s main data structure. It was originally created to be used as a practical mathematical notation for computer programs but became popular as a program for research in artificial intelligence.

There have been many versions of Lisp over the years, as well as many dialects. The most commonly used “regular Lisp” in use today is probably ANSI Common Lisp, of which there are also multiple implementations. To use ANSI Common Lisp on Ubuntu, install the package `clisp`. Type `clisp` from the command line to bring up a REPL (from which you may exit by entering `quit`).

Many Lisp programmers prefer to use `emacs` as their editor, which was written in a Lisp dialect called `elisp`. `emacs` includes many useful tools for Lisp and has other plug-ins available. From here it is easy to save code in files, compile it, and enable it to be run as programs rather than from the REPL interface.

Another interesting dialect of Lisp is Scheme, which is also available from the Ubuntu repositories but is not covered in this chapter.

Lua

Lua is a scripting language created in Brazil in the 1990s. It is similar to and based on Scheme. It is a dynamically typed procedural language with memory management and garbage collection. It is small and often used for embedded applications. It can be compiled on any platform that has a C compiler. Lua is also extensible, with a reputation for being simple without being simplistic. It was originally designed for extending applications but is frequently used for standalone and general-purpose needs.

To use Lua on Ubuntu, you write programs in your favorite text editor. To run them, you need the package `lua5.3`, which is the Lua interpreter. Run a program by entering `lua programName` at the command line.

Mono

Although Microsoft intended it for Windows, the Microsoft .NET platform has grown to encompass many other operating systems. No, this isn’t a rare sign of Microsoft letting customers choose which OS is best for them. Instead, the spread of .NET is because of the Mono project, which is a free reimplement of .NET available under the GPL license.

Because of the potential for patent complications, it took most distros a long time to incorporate Mono, but it's here now and works just fine. What's more, Mono supports both C# and Visual Basic .NET, as well as the complete .NET 1.0 and 1.1 Frameworks (and much of the 2.0 Framework, too), making it easy to learn and productive to use.

You can learn more about Mono from <https://mono-project.com>. In any case, to compile your own programs in Mono on Ubuntu, you need to install the `mono-devel` package.

OCaml

Functional programming never really goes away. Sometimes the most elegant way to write something is not by using a class or a method or a framework. Sometimes, the most elegant implementation is simply a function. This is why Lisp endures and why newer languages like OCaml and Haskell appear. Well, we say “appear,” but in reality OCaml is a modern dialect of a very old functional language called ML, which was developed in the early 1970s. OCaml is used primarily, but not exclusively, in the financial world, in programs for electronic trading, markets, and investments. It has an advanced type system and supports not only functional but also imperative and object-oriented styles of programming. It includes a memory manager and incremental garbage collection.

To use OCaml on Ubuntu, you write programs in your favorite text editor. To run them, you need the package `ocaml`, which includes two compilers: `ocamlc` compiles to bytecode, and `ocamlopt` compiles to native code.

Perl

Perl is a well-established programming language that has been around since the 1980s. It started as a Common Gateway Interface (CGI) language for web servers. Over time, people have used it for scripting, systems administration, network programming, and a ton of other things. You will find Perl everywhere being used in ways never dreamed of by its originators. It is an incredibly flexible and powerful language. The downside to this is that it is also a complex language that some jokingly describe as looking like a cat walked across your keyboard. If you know what you are doing with Perl, you can work magic. If you can't remember what you did and didn't document it, you will probably end up hating yourself—as will anyone else who has to interact with your code. All joking aside, it is worth learning, even just a little.

Perl is installed by default and already in use on your system. To use Perl on Ubuntu, you write programs in your favorite text editor. Nothing special is needed. Put the script somewhere and run it from the command line, like this: `perl yourscripename.pl`.

Note that there was a development effort underway to replace Perl 5. The new version was to be called Perl 6, but had diverged far enough away from Perl 5 that the decision was made to rename it to Raku. You can learn about Raku at www.raku.org.

This book has a digital-only chapter on the Perl language, available at www.informit.com/title/9780136778851.

PHP

PHP is another well-established programming language originally created for web development but that now sees use in many other roles. Often it is used for simple scripts on servers. PHP is kind of a cross between Java and Perl. It is quick and easy to learn and commonly found. PHP is another language worth learning, even if you learn just a little.

PHP is installed by default and already in use on your system. To use PHP on Ubuntu, you write programs in your favorite text editor. Nothing special is needed. Put the script somewhere and run it from the command line, like this: `php yourscripname.php`.

This book has a digital-only chapter on the PHP language, available at www.informit.com/title/9780136778851.

Python

Python is one of the easiest languages to read. It has been developed with the idea that there should be one obvious “right” way to do things. As a result, most people who use it believe that Python requires very few comments in the code because the code is easy to read and understand. For the most part, they are right. Python is also a powerful, fast, easy-to-use and easy-to-learn language that is worth learning, even if you learn just a little.

Python is installed by default and already in use on your system. To use Python on Ubuntu, you write programs in your favorite text editor. Nothing special is needed. Put the script somewhere and run it from the command line, like this: `python yourscripname.py`.

This book has a digital-only chapter on the Python language, available at www.informit.com/title/9780136778851.

Raku

Raku is the next evolution of Perl and was developed by people from the Perl community. Originally slated to become Perl 6, thus far it is primarily a specification with an early stage implementation available as Rakudo Star, which you can download as source at <https://rakudo.org/star/>.

Ruby

In Ruby, everything is an object. Every object can be given its own properties and methods. You can use closures (called *blocks* in Ruby). You do not need to declare variables, and only single inheritance exists. Ruby includes garbage collection and exception handling, and it can be extended by writing extensions in C. Ruby was heavily influenced in different ways by Lisp, Perl, Python, and Smalltalk and was originally designed for system administration-type scripting.

Most Ruby programmers seem to prefer using Ruby in combination with a web application framework called Rails, making what is known as *Ruby on Rails*. This framework is strongly tied to the DRY philosophy: “Don’t repeat yourself.” Every piece of information

is stored in a single, unambiguous place. Ruby on Rails runs on top of a web server like Apache or Nginx and is extensible using RubyGems (see <https://rubygems.org/>).

To use Ruby on Ubuntu, you write programs in your favorite text editor. To run them, you can install the interpreter package `ruby-full` from the Ubuntu repositories. Because Ruby changes often, the official Ruby documentation recommends *not* using a distribution's package manager but rather downloading the latest version directly from the Ruby website.

Rust

Rust is developed by Mozilla, the people behind the Firefox browser. It is advertised on www.rust-lang.org as follows:

...a curly-brace, block-structured expression language. It visually resembles the C language family, but differs significantly in syntactic and semantic details. Its design is oriented toward concerns of “programming in the large,” that is, of creating and maintaining boundaries—both abstract and operational—that preserve large-system integrity, availability and concurrency.

Rust is not yet available in the Ubuntu repositories, but as it is being developed and used by people at Mozilla, it is likely to be of interest to a few of this book's readers and deserves a quick mention here. It is also quickly catching up to the Go Language and may surpass it soon as the language of choice for large-scale web applications in the cloud.

Scala

Scala takes its name from “scalable language.” It is designed to grow with its users' needs. Scala runs on a JVM. It is suited for both functional and object-oriented programming. Scala programs are bytecode compatible with Java, and you can call either language from the other. Support for the .NET Framework is also available. Scala syntax is much more succinct than that of Java. Programs are generally shorter to write. As in Ruby, in Scala everything is an object. Types are inferred and do not need to be made explicit. Like Clojure, it suits the desire that many have to perform functional programming on a JVM.

To use Scala on Ubuntu, you write programs in your favorite text editor. To compile, you need the package `scala`. Use `scalac sourceFile` to compile, and to run using the interpreter, use `scala sourceFile`.

Scratch

Scratch is a programming language primarily designed for educators and children. It is from MIT and was created with the hope of making it easy to create fun interactive stories, animations, games, music, and art, all while teaching creative thinking, systematic reasoning skills, mathematical and computational ideas, and collaboration. Creations can be shared on the Web and then accessed from anywhere. Scratch has an online component, but development happens on a local machine.

To use Scratch on Ubuntu, install `scratch` and check out <https://scratch.mit.edu> to get started.

Vala

Vala is a very new language. It was designed to make the lives of the developers of GNOME easier by bringing features from modern languages into C for use in GNOME desktop environment development. The syntax is very similar to that of C#. Vala is a compiled language, but instead of being compiled directly to bytecode, Vala is compiled to C, which is then compiled with a C compiler for each specific platform.

In C, a programmer must manually manage references in memory. In Vala, this is automated if the built-in reference types are used instead of plain pointers. Vala also uses the GNOME GObject system to provide reference counting. For the most part, Vala is primarily used by people working on GNOME, which makes sense because this is the reason Vala was developed. Time will tell whether it receives wider interest.

To use Vala on Ubuntu, you write programs in your favorite text editor. To compile, you need the package `valac`, which is the Vala compiler. You then need to compile the output from that with a C compiler such as the GNU C compiler, described in Chapter 38, “Using Programming Tools.”

References

- ▶ www.adaic.org—The Ada Information Clearinghouse, an excellent resource for learning Ada
- ▶ <https://clojure.org>—The main website for Clojure
- ▶ <https://dlang.org>—The main website for D
- ▶ www.dartlang.org—The main website for Dart
- ▶ <https://elixir-lang.org>—The main website for Elixir
- ▶ <https://elm-lang.org>—The main website for Elm
- ▶ www.erlang.org—The main website for Erlang
- ▶ www.forth.org—The main website for the Forth Interest Group, a great place to learn more about Forth
- ▶ www.gnu.org/software/gforth/—The main website for Gforth, the GNU project’s implementation of Forth
- ▶ <https://gcc.gnu.org/fortran/>—The main website for Gfortran
- ▶ <https://groovy-lang.org>—The main website for Groovy
- ▶ <https://haskell.org>—The main website for Haskell
- ▶ www.java.com—The main website for Java

- ▶ **www.w3schools.com/js/default.asp**—The W3C Tutorial page for JavaScript, which is also a great place to learn HTML and CSS
- ▶ **<https://kotlinlang.org/>**—The main website for Kotlin
- ▶ **<https://lisp-lang.org/>**—The main website for Common Lisp
- ▶ **www.clisp.org**—The main website for GNU Clisp, an implementation of Common Lisp
- ▶ **www.lua.org**—The main website for Lua
- ▶ **www.perl.org**—The main website for Perl
- ▶ **<https://php.net>**—The main website for PHP
- ▶ **www.python.org**—The main website for Python
- ▶ **<https://raku.org>**—The main website for Raku
- ▶ **www.ruby-lang.org/en/**—The main website for Ruby
- ▶ **www.scala-lang.org**—The main website for Scala
- ▶ **<https://live.gnome.org/Vala>**—The main website for Vala
- ▶ **<https://scratch.mit.edu>**—The main website for Scratch
- ▶ **<https://help.ubuntu.com/community/PowerUsersProgramming>**—A wiki page about programming using Ubuntu as your development platform

CHAPTER 40

Helping with Ubuntu Development

This chapter focuses on becoming part of a larger team—specifically the group of talented men and women who develop and package software for Ubuntu as a whole. Ubuntu is made up of thousands of different components that are written in several different programming languages. Every component is available as source code, which is gathered into source packages. Source packages also include other content, specifically metadata describing things like copyright and licensing information, required dependencies, and build instructions for compiling or assembling the software for use. These source packages are used to build binary `.deb` packages, which are then included in the Ubuntu software repositories and made available for quick and easy installation by users. There are ways for anyone interested and willing to do a bit of study to use their skills to help.

When a bug is found, it gets fixed only if someone helps out and spends time exploring the program code to find and fix it. Then the fix becomes available to the wider community only if someone takes the time to build a new package that includes the fix and then uploads that package to an appropriate location in the Ubuntu software repositories.

This chapter has several parts. Each part builds upon the previous ones to give you a full understanding of the process. If only one part of the process interests you—say a later one, like packaging—that is okay. You will benefit from reading the previous sections of the chapter even if you only end up focusing your work on later steps in the process. You can participate in any part of the process provided that you are willing to learn how to do so. This chapter gets you started, but it is not the end of your

IN THIS CHAPTER

- ▶ Introduction to Ubuntu Development
- ▶ Setting Up Your Development System
- ▶ Fixing Bugs and Packaging
- ▶ References

required study. The process described in this chapter is accurate, but there are more interesting and intricate details that are not included here.

Software development, especially on a big project, is not something to be done lightly or flippantly. For that reason, patches and bug fixes are only accepted from people willing to put in the effort necessary to produce detailed work with the highest quality. Those who don't have the time or aren't interested in learning the steps properly will probably write software that reflects this, and they therefore will not help the community. The Ubuntu community is really nice, but they are also really busy. They will make time to mentor new developers and packagers who do their homework and make quality efforts, but it isn't polite to waste their time with half-hearted attempts at getting the details correct.

Introduction to Ubuntu Development

Ubuntu development follows a six-month cycle. The process begins with planning and discussion, with the goal of unifying the efforts of all those who will be involved to make the best use of limited resources. Developers get together via remote online participation, and in live discussions they make decisions about the priorities for the upcoming or the current release. These meetings happen once every three months, once at the start of a development cycle and once in the middle of the same cycle.

When a new release is created, it is given a new code name such as “Focal Fossa.” The release is referred to using this code name until its official release date, when it starts to be officially referred to using the release number, such as 20.04. See <https://wiki.ubuntu.com/Releases> for all the release names ever used by Ubuntu.

Some development begins by taking the newest versions of software from upstream software developers, either via Debian or directly from a specific application's source. In some cases modifications, or patches, are added to the code. All these modifications and patches that are added to the code are made into Ubuntu packages.

Other development begins internally, as some packages are written entirely by Ubuntu developers, including both those paid by Canonical and those who are volunteers in the wider Ubuntu community, doing so for personal or community benefit or just for fun. These packages receive the new features slated for the release and are then made into new Ubuntu packages.

Testing occurs throughout the cycle, starting as soon as the first new packages begin to be uploaded. Testing often uncovers bugs, which are problems with the software, such as errors that occur when the program is run. Bug reports are made using Launchpad, which is the issue tracker and code hosting solution created by Canonical for Ubuntu. Developers and community members read and triage bugs into categories based on their importance. Critical bugs must be fixed before final release. Important bugs should be fixed as soon as possible. Other bugs might be irritating but not vital to the operation of the program or the overall Ubuntu system, and, because of limited resources, they might be put off until someone has time to deal with them. Every time bugs are fixed, a new package must be made for the software so that it can be again uploaded into a repository for testing and review.

Alpha releases are made. Beta releases are made. Testing and bug fixing continue. Some software might be found to include new bugs that make the software unsuitable for use, so the previous version of the software may then be used. This is called “rolling back” to the previous version.

Eventually, the release day arrives. Everything works. The Ubuntu community rejoices and takes a week or two off. Then, it is time for the next set of planning meetings.

Setting Up Your Development System

Before you can begin helping with Ubuntu development, you must set up your system so that it works seamlessly with the rest of the Ubuntu community. There is a standard set of packages to install, and there are some standard steps you must perform.

Install Basic Packages and Configure

The Ubuntu development community has created a convenient package that installs everything you need to get started. Install `packaging-dev` from the Ubuntu software repositories, which among other things includes

- ▶ `gnupg`, the GNU Privacy Guard, which includes the cryptographic tools you need to sign your packages when you upload them to Launchpad
- ▶ `pbuilder`, which creates reproducible builds of a package in a clean, isolated environment
- ▶ `bzr-builddeb`, which includes `bzr`; together these serve as your version control and package management systems
- ▶ `apt-file`, which helps you find the binary package that includes a needed file

Set Up GPG

Generate your GPG key so that you can sign packages. Packages must be signed before they will be accepted for upload into the Ubuntu software repositories. This allows for tracking who is creating software and minimizes the risk of malicious software acts.

```
matthew@seymour:~$ gpg --gen-key
```

You are asked what kind of key you want to generate. You can safely choose the default settings. Create a passphrase when asked. When this is done, a message like this one is returned:

```
pub 2048R/38E0C789 2012-08-25

    Key fingerprint = 6363 387F 7455 8929 E6E2 4619 4798 DFD9 38E0 C789

uid Matthew Helmke <matthew@matthewhelmke.com>

sub 2048R/BDE097FF 2012-08-2
```

NOTE

I created this GPG key specifically for this book, and it is not used anywhere else. You won't see it in use on Launchpad, for example, or for signing packages.

You need the key ID, which in this example is `38E0C789`, and you need to upload your key ID to a keyserver, replacing `keyID` with your key ID:

```
matthew@seymour:~$ gpg --send-keys keyID
```

Creating Your SSH Key

Next, create an SSH key that enables you to connect securely to another computer for file transfer. This is used when you're uploading source files to Launchpad:

```
matthew@seymour:~$ ssh-keygen -t rsa
```

Choose the defaults and create a passphrase.

Setting Up pbuilder

Finally, set up `pbuilder` so that you can build packages on your local machine. Replace `release` with the name of the release for which you will develop (for example, the release being developed when this was written was `quantal`):

```
matthew@seymour:~$ pbuilder-dist release create
```

You can do this for multiple releases at the same time, and you can do it for Debian releases such as `sid` as well as for Ubuntu.

It takes some time for all the needed files to be downloaded and installed, but when they are, this step is complete.

Creating a Launchpad Account

Launchpad is described in Chapter 38, "Using Programming Tools." If you have already signed up for an account, you can skip this step. If not, open <https://help.launchpad.net/YourAccount/NewAccount> and sign up. You can develop for yourself without a Launchpad account, but to develop for Ubuntu, you must have an account.

Setting Up Your Environment to Work with Launchpad

Setting up your environment to work with Launchpad involves several steps that build on the steps you just completed.

Uploading Your GPG Key to Launchpad

Find your GPG fingerprint by running the following, replacing `youremail` with the email address you used earlier to create your GPG key:

```
matthew@seymour:~$ gpg --fingerprint youremail
```

Something like the following is returned:

```
pub 2048R/38E0C789 2012-08-25

    Key fingerprint = 6363 387F 7455 8929 E6E2 4619 4798 DFD9 38E0 C789

uid                Matthew Helmke <matthew@matthewhelmke.com>

sub 2048R/BDE097FF 2013-08-25
```

Open <https://launchpad.net/~/+editpgpkeys> and copy the entire set of numbers and letters to the right of `Key fingerprint =` into the text box on the web page. Click Import Key on the web page. Launchpad then uses this fingerprint to check the Ubuntu key server for the key that you uploaded earlier.

You are sent an encrypted email and asked to confirm the key import. You need to use an email client that supports OpenPGP encryption and enter the passphrase you created when you created the key in order to read this email. If you do not have an email reader that supports OpenPGP, you can also perform this step at the command line by typing `gpg` and copying/pasting the text of the email into the terminal window before pressing Enter.

Follow the directions in the email to complete this step.

Uploading Your SSH Key to Launchpad

Open <https://launchpad.net/~/+editsshkeys>. Also open `~/.ssh/id_rsa.pub` in a text editor. The contents of this file are the public part of your SSH key, so it is safe to share it with Launchpad. Copy the contents of the file and paste them into the Add an SSH Key text box on the web page. Then click Import Public Key.

Configuring Bazaar

Bazaar is covered in Chapter 38, “Using Programming Tools.” Here, all you need to do is tell Bazaar you who are. This is a two-step process. The first step is to tell Bazaar which name and email address it should use when creating commit messages. Replace my name and email address with your information in the following command:

```
matthew@seymour:~$ bazaar whoami "Matthew Helmke" <matthew@matthewhelmke.com>
```

The second step is to set up Bazaar with your Launchpad ID so that the code you sign and upload is associated with your Launchpad account. Replace *yourLaunchpadID* with your Launchpad ID in the command shown here:

```
matthew@seymour:~$ bazaar launchpad-login yourLaunchpadID
```

If you can't remember your ID, see where <https://launchpad.net/~> redirects you. The part that is automatically added after the `~` is your Launchpad ID.

Configuring Your Local Bash Shell

The Ubuntu packaging tools that run at the command line on your development machine need to be configured with your information as well, in the same way that they are configured for Ubuntu's parent, Debian. Open `~/.bashrc` in your favorite text editor and add the following lines at the end, changing them to use your information:

```
matthew@seymour:~$ gpg --fingerprint youremail
```

Reload the Bash shell configuration file as shown here, again making the changes needed to use your own information:

```
matthew@seymour:~$ export DEBFULLNAME="Matthew Helmke"
matthew@seymour:~$ export DEBEMAIL="matthew@matthewhelmke.com"
```

If you are using a shell other than the default, Bash, you need to configure your shell similarly.

Fixing Bugs and Packaging

This section covers the process for fixing bugs and packaging your code. It does not cover the mechanics of reading program code and fixing it. Here, it is assumed that you know how to program in the language used in the software you are fixing.

From a high level, the process is easy to understand. Here are the steps:

1. Find a problem (a software bug).
2. Download the source code.
3. Fix the problem.
4. Test the fix on your local machine.
5. Commit the changes.
6. Request that your changes be merged into the main source.

Finding problems to fix is something you learn with time and experience. If you aren't fixing an issue you discovered yourself, other places you can look are in Ubuntu mailing lists, Launchpad bug reports, and community gathering places such as the Ubuntu Forums or AskUbuntu.

Before you do any work on a bug, do an extensive search to make sure it has not already been fixed and that someone else is not already working on the bug. You should obviously look in Launchpad, in the section for the specific package. You can also check the upstream and/or Debian bug trackers for open and closed bugs and the upstream revision history or newer release(s). When you know of a bug you want to fix, download the source code from the Ubuntu software repositories. Find out which file contains the code you want to work on. The example in this section uses an imaginary application, `matthewsapp`. In all the examples that follow, replace `matthewsapp` with the name

of the program or application that you want to work on. Say that you know that the `matthewsapp` binary is located at `/usr/bin/matthewsapp`. To find the Ubuntu package that contains `matthewsapp`, enter the following:

```
matthew@seymour:~$ apt-file find /usr/bin/matthewsapp
```

This returns output something like the following.

```
matthewsapp: /usr/bin/matthewsapp
```

You can also use `apt-cache` to learn the source package for a binary package. Sometimes source packages have the same name as their resulting binary packages, and sometimes they do not. Here are two examples.

```
matthew@seymour:~$ apt-cache showsrc tomboy | grep ^Package:
```

```
Package: tomboy
```

```
matthew@seymour:~$ apt-cache showsrc python-vigra | grep ^Package:
```

```
Package: libvigraimpex
```

In the first example, the binary `tomboy` is in the `tomboy` package. In the second example, the binary `python-vigra` is in the `libvigraimpex` package. When you know the package that contains the source code you need, get a copy of the source code itself. You do this in Ubuntu by branching the source package. Launchpad manages all the code for source packages and is the place from which you will download and create a local branch. You fix bugs in the local branch and test them on your machine and then submit a merge proposal so that code from your branch can be examined and merged back into the main branch of the code to then be used to build a new package of the code.

Create a local repository on your machine for the code, again replacing `matthewsapp` with the name of the source package you need:

```
matthew@seymour:~$ bzip init-repo matthewsapp
```

Change to the newly created directory:

```
matthew@seymour:~$ cd matthewsapp
```

Create a new local branch, and name it something obvious. Many Ubuntu developers name the new target directory the same as the original, with `.dev` added at the end, like this:

```
matthew@seymour:~$ bzip branch ubuntu:matthewsapp matthewsapp.dev
```

After you have done this a few times with packages already in the Ubuntu repositories, you can read the official Ubuntu Packaging Guide to find out how to perform similar tasks with code from other Ubuntu releases, from Debian, or from an upstream `.tar` file. This section concentrates on doing this the easy way to help you get started.

Create a patch that will include the fix for the bug:

```
matthew@seymour:~$ edit-patch 99-new-patch
```

This copies the packaging of the file to a new temporary directory. Edit the files with a text editor or use a patch to do so in a temporary shell, like this:

```
matthew@seymour:~$ patch -p1 < ../bugfix.patch
```

You can exit the temporary shell by entering `exit` or using `Ctrl+D`.

Build a test package using your patch to test your changes, replacing `release` with the name of the release, such as `quantal`, and replacing `package` and `version` with the name and number of the package:

```
matthew@seymour:~$ bzip builddeb -- -S -us -uc
matthew@seymour:~$ pbuilder-dist release build ../package_version.dsc
```

When the build completes, install the package from `~/pbuilder/release_result/` and test to see if the bug is fixed:

```
matthew@seymour:~$ sudo dpkg -i package_version.deb
```

Test it. Get it running. Try to break it. If you are convinced the bug is fixed in the software, only then is it time to move on. Feel free to repeat this cycle as often as necessary until it is really fixed.

When your changes are complete, create a new entry in the `debian/changelog` file:

```
matthew@seymour:~$ dch -i
```

Boilerplate text for the `changelog` entry that includes the first and last lines with placeholder text for the middle is provided for you. Use a text editor to edit the middle line(s) and include in your entry a specific bug fix tag that indicates which Launchpad bug you are fixing. Make sure you include where in the code you made the change(s), what you changed, and where the discussion of the change occurred.

The format for this is quite strict and looks as follows:

```
matthewsapp (0.9.2-1ubuntu3) quantal; urgency=low
```

```

 * debian/control: don't bacon the narwhals at midnight in line 35 as
 discussed on launchpad (LP: #3263827)
```

```
-- Matthew Helmke <matthew@matthewhelmke.com> Sat, 25 Aug 2013 13:29:01 -0500
```

Commit the change locally:

```
matthew@seymour:~$ debcommit
```

Push the change to Launchpad, replacing items in *italic* with your information:

```
matthew@seymour:~$ bzr push lp:~yourLaunchpadID/ubuntu/release/package/branchname  
matthew@seymour:~$ bzr lp-propose
```

The first command sends your code to Launchpad. The last command opens the Launchpad page of the merge proposal in your browser.

References

- ▶ <https://developer.ubuntu.com>—The main website for Ubuntu development.
- ▶ <https://packaging.ubuntu.com>—The main website for the Ubuntu Packaging Guide, which helped greatly in the writing of this chapter and goes into far greater detail than this chapter. (This packaging guide is also available directly from the Ubuntu software repositories in the `ubuntu-packaging-guide` package.)

This page intentionally left blank

CHAPTER 41

Helping with Ubuntu Testing and QA

IN THIS CHAPTER

- ▶ Community Teams
- ▶ Bug Squad
- ▶ References

There are many ways to help the Ubuntu community create, refine, and promote the operating system. Some are highly technical, like writing code or packaging programs to be included in the software repositories. Some are less technical, such as helping promote Ubuntu locally or through blogging interesting news items from the community. Somewhere in the middle, leaning toward the technical side, is a task that is wide open for greater community involvement.

This is a rather brief and intentionally vague chapter. Testing by volunteers requires more than a casual interest if it is going to be helpful to the developers and not an annoyance. For that reason, this chapter covers the basics of how to get involved and some of the opportunities but not the precise details. If you are interested—and, after reading this chapter, we hope you are—the next step is to visit the websites listed in the “References” section at the end of the chapter.

Community Teams

Two community teams—the Ubuntu Testing Team and the Ubuntu QA Team—would love to see volunteers who can follow directions, be careful and methodical, and notice details. Both teams work to refine the distribution during the development cycle to help make Ubuntu the best it can be.

The Ubuntu QA Team looks directly at the overall quality of the distribution, trying out default programs and configurations and trying to break things. The goal is to find bugs during the development cycle and make clear reports about them, trying to get them fixed before end users ever know of their existence. The goal here is product improvement and quality control or assurance.

These are big tasks. Even with several Canonical employees working on these tasks full time, testing every hardware configuration or use case is nearly impossible. Guessing what creative ways users will attempt to perform tasks that developers have designed to perform differently is also impossible. Testing as many of these options as possible is key when the goal is to create a positive experience for as many people as possible. This is why volunteers are both welcome and actively recruited.

These are also exacting tasks. Testing and bug reporting (or fixing) require careful attention so that problems are reported clearly with steps that can be repeated by developers. This enables them to find where problems lie and more easily fix them. Not everyone is well suited for this sort of thing, but those who are able to be clear and precise and who can follow the directions given in testing plans and procedures are worth their weight in gold. You might not receive public glory for testing, but you will receive honor from those working with you in testing if you can do the job well.

You will notice some crossover in the descriptions of the teams that follow because these teams work together closely. Although each team has a main focus, the two teams perform similar tasks from time to time. Even the individual teams' web pages link to each other and offer similar information to help coordinate and direct any interested volunteers to the tasks they feel most equipped to help with.

Ubuntu Testing Team

Members of the Ubuntu Testing Team are probably best known for their work during a release week, when they help validate all the CD and DVD images. They also operate on the release cycle testing beta releases and release candidates. In addition, they test update packages for stable releases before the packages are pushed out to users. They do this by enabling a new software repository called `-proposed` and trying out the software there before it ends up in the `-updates` repository. In addition, they help developers by coordinating communications and actively seeking and connecting additional volunteers when specific testing needs are encountered.

To join the Ubuntu Testing Team, you need a Launchpad account, and then you can join the team via the team's Launchpad page, at <https://launchpad.net/~ubuntu-testing>. Subscribing to the team's email list at <https://lists.ubuntu.com/mailman/listinfo/ubuntu-qa> is also a good idea. When testers are needed, the Launchpad team list and the mailing list are the initial points of contact for those who are asked to help.

There are many ways to get involved in testing; each has varying requirements in terms of time and technical skill. Some tasks are quick and easy, and others are more involved. Some of the tests are general in nature, and others involve specific features, applications, and hardware. Some tests are automated and require specific test software to be downloaded and installed. Others are given as a list of instructions. Opportunities exist for many skill levels and most time schedules.

Ubuntu QA Team

Compared to the Ubuntu Testing Team, the QA Team has a stronger focus on developing and using tools to automate the process. This enables people using the tools to run a

large battery of tests against a code base very quickly while ensuring that they are probing precisely what is needed. Much of the QA Team's work is useful for hardware certification, logic testing, and bug discovery. The team has developed an Ubuntu developer-focused suite of tools called `ubuntu-qa-tools`, a library called Mago for the *Linux Desktop Testing Project (LTP)* to simplify testing of Ubuntu within the wider realm of Linux desktops, a framework called Checkbox that tests and sends test data directly to Launchpad, and more.

Bug Squad

One of the first places people become involved in Ubuntu testing is with Bug Squad. These volunteers are the initial point of contact for most bugs filed with Ubuntu. They read bug reports to see whether each bug seems legitimate (for example, that it isn't something like "Ubuntu doesn't work" but is specific and measurable), that it is filed against the appropriate software package, and that adequate information is included with each report for the developers to be able to figure out the problem. If the bug report meets the criteria, the Bug Squad determines which developer or team should be notified. Sometimes during the triage process, the Bug Squad might ask reporters for more information. The volunteers attempt to determine the severity of the bug and might assign a priority setting to the report.

References

- ▶ <https://wiki.ubuntu.com/QATeam>—The starting point for understanding and volunteering for QA Testing
- ▶ <https://wiki.ubuntu.com/BugSquad>—The Ubuntu Bug Squad

This page intentionally left blank

Index

Symbols

- & operator, 237
 - background processes, running, 173–174
 - special shell character, 247
 - VirtualBox commands, 527
- && operators, combining commands, 180, 198
- * special shell character, 247
- \ special shell character, 249–250
- ` special shell character, 247, 250
- [] special shell character, 248
- [a, z] special shell character, 248
- [a-z] special shell character, 248
- [az] special shell character, 248
- \$, 235
- \$? built-in variable, 247
- \$ special shell character, 247
- \$# built-in variable, 246
- \$* built-in variable, 247
- \$0 built-in variable, 247
- !! operator, 196–197
- != operator, 491
- < special shell character, 247
- << operator, 236
- << special shell character, 248
- > special shell character, 247
- >> special shell character, 247
- | (pipe) special shell character, 247
- || operator, 198, 491
- # special shell character, 247
- ? special shell character, 247

650 " special shell character

" special shell character, 248–249

' special shell character, 249

Numbers

10BASE-T networking, 339

10G Ethernet networking, 340

32-bit Ubuntu, 64-bit Ubuntu versus, 4–5

50G Ethernet networking, 340

64-bit Ubuntu, 32-bit Ubuntu versus, 4–5

100BASE-T networking, 339

1000BASE-T networking, 339

A

A records, 590

AAAA records, 590

ac command, 211, 223

accessing

ACL

setting permissions, 127–128

Squid, 565–569

Apache web server, 452, 455

command line, 107

databases

local GUI clients, 503

PostgreSQL, 501–503

SSH, 501–502

web browsers, 503

file access time, disabling, 397

networks, chains, 387

remote access

copying files between machines,
372–373

Guacamole, 377

key-based logins, 373–375

online references, 377–378

SSH server, 371–375

VNC, 375–377

shell scripts, storing for system-wide access,
240

variable values, 243

accounting, user usage statistics, 212

ACID compliance, relational databases,
493–494

ACL

permissions, setting, 127–128

Squid, 565–569

activity (user), monitoring, 211–212

Ada, 622

addressing

broadcasting, 338

IPv4 addressing, 331–332

IPv6 addressing, 332, 334–336

multicasting, 338

NAT, 332

TCP/IP, 331–332

unicast addressing, 338

adduser command, 136–137, 209

administration

LDAP, 584–585

privileges, 217

Adobe Photoshop, 67

afio, 316

all-in-one (Print/Fax/Scan) devices, 435

AllowOverrides directives (Apache web server),
451–452

ALSA, 64

Amanda, 315–316

AND, 299

AND operators, 491–492

Android mobile development, 607

Android Runtime, 608

Android Studio, 609

- Application Framework, 608–609
- applications, 609
- ARM processors, 608
- Google Play, 607
- Linux kernels, 608
- RISC processors, 608
- Ansible, 544
- Apache Tomcat, 482
- Apache web server
 - access control, 452, 455
 - configuring
 - .htaccess configuration files, 450–452, 469
 - runtime servers, 446–449
 - threads, 468
 - development of, 443–444
 - file system authentication, 452–455
 - .htaccess configuration files, 450, 469
 - AllowOverrides directives, 451–452
 - Options directives, 451
 - HTTPS, 464–466
 - installing, 444
 - logging, 463–464
 - modules, 455–461
 - MPM, 449–450
 - online references, 466
 - overview of, 444
 - Require directive, 452
 - runtime servers
 - apache2.conf files, 447–449
 - configuring, 446–449
 - DirectoryIndex directive, 449
 - DocumentRoot directive, 449
 - GID, 448
 - Group directive, 448
 - Listen directive, 447
 - ServerAdmin directive, 448
 - ServerName directive, 448–449
 - ServerRoot directive, 447
 - UID, 448
 - User directive, 448
 - UserDir directive, 449
 - security, 444, 452–455
 - starting/stopping, 444–446
 - threads, 468
 - virtual hosting, 461–462
- apache2.conf files, 447–449
- API, 512
- AppArmor, 388–390
- Application Framework, 608–609
- applications
 - Android applications, 609
 - AppArmor, 388–390
 - CD/DVD burning applications
 - Brasero, 71
 - Linux command line, CD creation, 72–73
 - Linux command line, DVD creation, 73–75
 - Ubuntu support, 71
 - DVD/CD burning applications
 - Brasero, 71
 - Linux command line, CD creation, 72–73
 - Linux command line, DVD creation, 73–75
 - Ubuntu support, 71
 - games, 79–81
 - Battle for Wesnoth, 85–86
 - commercial games, 88
 - documentation, 89
 - emulators, 79
 - FlightGear, 87
 - Frets on Fire, 86
 - Frozen Bubble, 84
 - Game Jolt, 82
 - GOG.com, 82
 - Humble, 82

- installing from Ubuntu repositories, 82–87
- installing video drivers (proprietary), 80
- itch.io, 82
- kid-friendly games, 88
- LGDB, 82
- online game sources, 81–82
- Scorched 3D, 83–84
- Speed Dreams, 87
- SuperTux, 84–85
- Warsow, 82
- Windows games, 88–89
- graphics manipulation applications, 66, 70
 - Blender, 70, 77
 - CinePaint, 70
 - darktable, 70
 - digikam, 70
 - GIMP, 66–67
 - Hugin, 70
 - Inkscape, 70
 - Krita, 70
 - nautilus-image converter, 69
 - netpbm tools, 69
 - Photoshop, 67
 - POV-Ray, 70
 - Radiance, 70
 - Shotwell Photo Manager, 66, 71
 - Simple Scan, 67
 - Xara Xtreme, 70
- microservice architectures, 531
- multimedia applications
 - ALSA, 64
 - CD/DVD burning applications, 71–75
 - graphics manipulation applications, 66–71
 - music applications, 65–66
 - online references, 77
 - OSS, 64
 - PulseAudio, 64
 - sound cards, 63–64
- music applications, 65
 - Banshee, 66
 - Rhythmbox, 66
 - Sound Juicer, 66
- productivity applications
 - Celtx, 59
 - CrossOver Office, 61
 - defined, 56
 - gedit, 59
 - Heimer, 61
 - Kile, 61
 - LaTeX, 60
 - LibreOffice, 55, 56–58
 - LyX, 60
 - online references, 61
 - pdfedit, 58
 - Publican, 59–60
 - Texmaker, 60
 - users (typical), 56
 - Windows, 61
 - Wine, 61
 - XML Copy Editor, 60
- recording/editing sound, 76
- security, 388–390
- Startup Applications Preferences, 278
- VBA, LibreOffice and Office compatibility, 55
- video editing applications, 77
 - Avidemux, 77
 - Blender, 77
 - Cinelerra, 77
 - DaVinci Resolve, 77
 - Kdenlive, 77
 - Lightworks, 77
 - OpenShot Video Editor, 77
 - PiTiVi, 77
 - Shotcut, 77

apropos command, 112
 APT, 95

- apt-get versus, 99–100
- day-to-day usage, 95–98
- finding software, 98–99
- VirtualBox, 527

 apt-get, APT versus, 99–100
 archives, restoring files from, 311–312
 Ardour, 76
 arguments (positional), 242
 ARM processors, 608
 arrays (RAID), 307
Art of Unix Programming, The, 141–142
 ash, 226
 Aslett, Matt, 511
 assessing

- needs/resources, backup strategies, 303–304
- vulnerabilities, 381–382

 assigning

- permissions, file systems, 121–122
- values to variables, 242

 at command, 227–228, 229
 atomic changes, 613
 attacks, 380–381

- autocracking scripts, 380
- crackers, 380
- external attacks, 379–380
- hackers, 380
- internal attacks, 379–380
- script kiddies, 380
- spoofing attacks, 593
- viruses, 385
- war driving, 383
- worms, 380

 Audacity, 76

Audio CD Extractor. See Sound Juicer
 authentication

- file systems, Apache web server, 452–455
- PAM, 215–216

 autoconf command, 601–602
 autocracking scripts, 380
 automatically waking computers from sleep, 231–233
 automating tasks with scripts, 244–246
 availability

- localhost interfaces, checking availability of, 326
- memory space, displaying, 286

 .avi files, 75
 Avidemux, 77
 awk text editor, 189–191
 AWS, 539
 Azure, 539

B

Back In Time, 314
 background jobs, moving jobs to, 174–175
 background processes, 173–174 237

- running,

 backups

- afio, 316
- Amanda, 315–316
- Back In Time, 314
- cloud computing, 305–306, 309
- configuration files, version control, 320–322
- copying files, 316–317
 - cp command, 318
 - rsync command, 319–320
 - tar command, 317–318
- data loss, 302
- Déjà Dup, 313–314
- external hard drives, 308

- File Roller, 312
- flexbackup, 316
- full backups
 - with incremental backups, 307
 - on a periodic basis, 306–307
 - tar command, 310–311
- incremental backups, 310–311
- kdat, 312–313
- KDE ark archiving tool, 312–313
- levels of, 306
- mirroring data, 307
- NAS, 308
- network storage, 308
- online references, 324
- restoring files from archives, 311–312
- scheduling tasks, full backups on a periodic basis, 306–307
- software, 309–316
- strategies, 301
 - assessing needs/resources, 303–304
 - choosing a strategy, 308
 - choosing hardware/media, 308–309
 - evaluating, 304–306
 - full backups on a periodic basis, 306–307
 - full backups with incremental backups, 307
 - home users, 304–305
 - incremental backups, 307
 - inheriting, 306
 - large enterprises, 305
 - RAID arrays, 307
 - simple backup strategies, 306
 - small enterprises, 305
 - small offices, 305
- system rescue, 323
 - restoring GRUB2 boot loaders, 323–324
 - saving files from nonbooting hard disks, 324
 - Ubuntu Rescue disc, 323
 - tape drive backups, 309
 - tar command, 310
 - copying files, 317–318
 - incremental backups, 310–311
 - restoring files from archives, 311–312
 - Ubuntu installations, 7
 - Unison, 315
 - version control, configuration files, 320–322
- badblocks command, 397
- Banshee music application, 66
- BaseX, 517
- bash, 226
 - comparisons of expressions, 250–251
 - file operators, 253–254
 - logical operators, 255
 - numeric comparisons, 252–253
 - strings, 251–252
 - local bash shells, configuring, 640
- batch command, 228–229
- batches, changing passwords in, 216–217
- Battle for Wesnoth, 85–86
- Bazaar
 - configuring, 639
 - software project management, 611–612
- Berkeley DB, 512, 579
- bg command, 174–175
- BigTable, 517
- Bikeshed, 616–618
- /bin directory, 114
- BIND
 - configuring DNS servers, 593–595
 - online references, 595
- BIOS, 271
 - boot process, 272–274
 - Ubuntu bootups, troubleshooting, 195
- Blender
 - graphics manipulation, 70
 - video editing, 77

- .bmp files, 68
 - BOFH, 205
 - boolean operators, combining commands, 180
 - boot loaders, 271
 - boot process, 272–274
 - GRUB2, installing, 5
 - GRUB2 boot loaders, restoring, 323–324
 - Ubuntu installations, 2–3, 5–6
 - boot process
 - BIOS, 271, 272–274
 - boot loaders, 271, 272–274
 - Boot-Repair, 278–279
 - DHCP activation, 357
 - GPT, 274
 - init systems, 272
 - loading process, 272–274
 - MBR, 274
 - NetBoot, 274
 - nonbooting hard disks, saving files from, 324
 - PXE, 274
 - runlevels, 271–272, 275, 278
 - services
 - controlling, 278
 - running, 271–272
 - starting/stopping, 275–278
 - Startup Applications Preferences, 278
 - troubleshooting, 278–279
 - Ubuntu, troubleshooting, 195–196
 - UEFI, 273–274
 - Brasero, 71
 - break statements, 269
 - bridges, 343, 524–525
 - broadcasting, 338
 - browsers (web)
 - Chrome, 53–54
 - Chromium, 53–54
 - database access, 503
 - Firefox, 52–53
 - online references, 54
 - brute-forcing, 373–374
 - Budgie, 49
 - bugs, fixing
 - Bug Squad, 647
 - Ubuntu development, 640–643
 - built-in variables, 242, 246–247
 - bundles (software), Snappy package manager, 103–104
 - bunzip2 command, 191
 - burning CD/DVD
 - Brasero, 71
 - Linux command line
 - CD creation, 72–73
 - DVD creation, 73–75
 - Ubuntu support, 71
 - business applications, Ubuntu for, 27–28
 - byobu, multiple terminal operation, 192–193
 - bzip2 command, 191
- ## C
- C programming, 597–599
 - autoconf command, 601–602
 - building programs, 599–601
 - configuring code, 601–602
 - debugging, 602
 - checking source code, 602–603
 - gdb command, 603
 - gprof command, 603
 - splint command, 602–603
 - symbolic debugging, 603
 - tracking function time, 603
 - dependency checking, 599–600
 - GCC, 603–604
 - macros, 600–601

- makefile targets, 600–601
- makefiles, 599–600
- C++ programming, 597–598, 599
 - autoconf command, 601–602
 - building programs, 599–601
 - configuring code, 601–602
 - debugging, 602
 - checking source code, 602–603
 - gdb command, 603
 - gprof command, 603
 - splint command, 602–603
 - symbolic debugging, 603
 - tracking function time, 603
 - dependency checking, 599–600
 - GCC, 603–604
 - macros, 600–601
 - makefile targets, 600–601
 - makefiles, 599–600
- C10K problem, 467–468
- cabling, 340
 - fiber-optic cabling, 342
 - UTP cabling, 341
- Caddy, 482
- cameras (digital)
 - Ubuntu support, 70
 - USB connections, 70
- capturing screen images, 69
- case statements, 267–268
- Cassandra, 513
- cat command, 132, 144–145
- /cd command, 120
- cd command, 145–147
- CD tools, 613–614
- CD/DVD burning applications
 - Brasero, 71
 - Linux command line
 - CD creation, 72–73
 - DVD creation, 73–75
 - Ubuntu support, 71
- cdrecord command, 72–73
- Cecilia, 76
- Celtx, 59
- CFEngine, 545
- cgroups, 532
- chains, 387
- change command, 223
- changing
 - desktop environments, 39
 - directories, 120, 145–147
 - passwords in batches, 216–217
 - permissions, files/folders, 125, 147
 - shells, 226
 - user identities, 217–219
 - window managers, 39
- Chaos Engineering, 614
- Chef, 544
- Cherokee, 480–481
- chfn command, 224
- chgrp command, 125, 204, 224
- children directories, 118
- children, games for, 88
- chmod command, 123–124, 147, 204, 224
- chown command, 125, 204, 224
- chpasswd command, 224
- Chrome, 53–54
- Chromium, 53–54
- chroot jail, 531–532
- chsh command, 209, 224
- Cl/CD tools, 613–614
- Cinelerra, 77
- CinePaint, 70
- CLI. See command line
- client/server database model, 484
- Clojure, 622–623
- cloud computing
 - AWS, 539
 - Azure, 539
 - backups, 305–306, 309

- containers, 531
- DevOps, 535–536
- Google Cloud, 539
- hybrid clouds, 539–540
- laaS, 537
- Juju, 540–541, 545
- Landscape, 541, 545
- MaaS, 537
- Mojo, 541
- online references, 541
- OpenStack, 538–539
- PaaS, 537
- private clouds, 540
- public clouds, 538, 539
- reasons for using, 536–537
- SaaS, 537
- Sysadmin, 535–536
- Ubuntu, 535
 - AWS, 539
 - Azure, 539
 - considerations for using, 538
 - Google Cloud, 539
 - hybrid clouds, 539–540
 - laaS, 537
 - install instructions, 536
 - Juju, 540–541, 545
 - Landscape, 541
 - MaaS, 537
 - Mojo, 541
 - OpenStack, 538–539
 - PaaS, 537
 - private clouds, 540
 - public clouds, 538, 539
 - reasons for using, 536–537
 - SaaS, 537
 - Ubuntu Cloud, 536
 - VM, 536
- VM, 536
- CNAME records, 590–591
- COBOL, 623–624
- code packaging, Ubuntu development, 640–643
- combining commands
 - && operators, 180, 198
 - !! operator, 196–197
 - || operator, 198
 - boolean operators, 142–143
 - pipng, 142–143, 178–180
- comm command, 170–171
- command line (Linux)
 - !! operator, 196–197
 - || operator, 198
 - accessing, 107
 - apropos command, 112
 - APT, 95
 - apt-get versus, 99–100
 - day-to-day usage, 95–98
 - finding software, 98–99
 - background jobs, 174–175
 - background processes, running, 173–174
 - basic commands, 143–144
 - BIOS, troubleshooting Ubuntu bootups, 195
 - byobu, multiple terminal operation, 192–193
 - CD creation, 72–73
 - combining commands
 - && operators, 180, 198
 - !! operator, 196–197
 - || operator, 198
 - boolean operators, 180
 - pipng, 178–180
 - command history, viewing, 197
 - commands
 - combining, 178–180, 196–197, 198
 - list of, 143–144
 - redirecting input/output of, 167–170
 - shortcuts, 198
 - standard input/output, 169

- commonly used commands/programs, 139
- Coreutils, 199
- defined, 106–107
- directories, 112–114
 - /bin directory, 114
 - changing, 120, 145–147
 - children directories, 118
 - confining scripts to directories, 198–199
 - copying files, 131, 147
 - creating, 129, 160
 - creating files, 128–129
 - deleting, 129–131, 161–162
 - displaying contents of files, 132
 - /etc directory, 114
 - finding current directory, 120
 - finding files, 147
 - /home directory, 115
 - listing contents of, 118–119
 - listing files in directories, 156–158
 - moving files, 131, 161
 - parent directories, 118
 - permissions, 122–127
 - printing directory sizes, 148
 - printing last lines of files, 163–164
 - /proc directory, 115–117
 - regular expressions, 133
 - renaming files, 131, 161
 - /sbin directory, 114
 - sorting file contents, 162–163
 - /tmp directories, 117
 - /usr directory, 117
 - /var directory, 117
 - wildcard characters, 133
- documentation
 - manual pages, 111–112
 - reading, 111–112
- DVD creation, 73–74
 - packet writing, 74–75
 - session writing, 74
- environment variables, 182–185
- file systems
 - directories, 112–117
 - hierarchy of, 112–114
- files/folders
 - comparisons, 170
 - compressed files, 191–192
 - creating links, 154–156
 - downloading, 164
 - finding differences in files, 170
 - finding files, 149–151
 - finding from indexes, 156
 - finding similarities in files, 170–171
 - listing in directories, 156–158
 - listing system information, 158–159
 - printing contents of, 144–145
- foreground jobs, 174–175
- GRUB, troubleshooting Ubuntu bootups, 195–196
- input string searches, 151–152
- jobs
 - executing in parallel, 181–182
 - listing, 173
- kernel ring buffers, reading contents of, 200
- location of commands, printing, 164
- logging out of, 108
- logins
 - from remote computers, 108–109
 - text-based logins, 107–108
- manual pages, 159–160
- multiple terminal operation, 192–193
- MySQL command line client, 504–505
- network interface configuration
 - ifconfig command, 346–348
 - ip command, 348–349

- ip route command, 346, 349
 - netstat command, 346, 350
 - online references, 139, 165, 200
 - paging through output, 152–154
 - pipng commands, 178–180
 - polite system resets, 194–195
 - PostgreSQL command line client, 505–506
 - printing, resource usage, 175–177
 - processes
 - listing, 171–173
 - prioritizing, 177–178
 - substituting, 181
 - reasons for using, 142–143
 - REISUB, 194–195
 - repeating text, 148–149
 - rerunning previous commands, 196–197
 - resource usage, printing, 175–177
 - root accounts, 110–111, 133–134
 - sequence, running commands in, 180–181
 - sudo command, troubleshooting, 134–136
 - systems
 - rebooting, 138–139
 - shutting down, 137–138
 - text editors, 185–191
 - Ubuntu bootups, troubleshooting, 195
 - user accounts
 - creating, 136–137
 - deleting, 137
 - super user accounts, 109–111
 - whereis command, 112
- command line (shells), 233–234
- background processes, 237
 - pattern-matching, 235
 - pipng commands, 237
- commands
- & operator, running background processes, 173–174
 - && operators, 180, 198
 - ! ! operator, 196–197
 - || operator, 198
 - database commands, 506
 - history of commands, viewing, 197
 - pipng, 142–143, 178–180, 237
 - printing location of, 164
 - redirecting input/output, 167–169
 - redirecting input/output with shell command line, 236
 - rerunning previous commands, 196–197
 - running, in sequence, 180–181
 - shortcuts, 198
 - substituting, 250
 - wildcard characters, 235
- commercial games, 88
- community teams, testing Ubuntu, 645–646
- comparing files/folders, 170
- comparisons of expressions, 250–251
- file operators, 253–254, 257–258
 - logical operators, 255, 259
 - numeric comparisons, 252–253, 256–257
 - strings, 251–252, 255–256
- compatibility
- free/open-source software, MS Office compatibility, 55
 - LibreOffice, Office compatibility, 55
 - Office, free/open-source software compatibility, 55
 - open-source/free software, MS Office compatibility, 55
 - Ubuntu installations, Mac hardware compatibility, 3
- compiling
- kernels, 412–414, 418–419
 - software
 - from source, 100, 101–102
 - from tarballs, 100–101
 - from Ubuntu repositories, 101–102
- compressed files, 191–192, 318

- computers (remote), command line logins, 108–109
- configuration files
 - Dotfiles, 322
 - version control, 320–322
- configuring
 - Apache web server
 - .htaccess configuration files, 450–452, 469
 - runtime servers, 446–449
 - threads, 468
 - bash shells, 640
 - Bazaar, 639
 - C/C++ code, 601–602
 - configuration management, 102
 - dotdee, 102–103
 - Snappy package manager, 103–104
 - Ubuntu Core, 103
 - DHCP software, 358–359
 - dial-up Internet access, 367–368
 - disk quotas, 223
 - DNS servers with BIND, 593–595
 - DSL Internet access, 365
 - /etc directory configuration files, 114
 - firewalls, 386–388
 - GPG keys, 637–639
 - hosts (networks) with DHCP, 359–361
 - Internet connections, common configuration information, 364–365
 - kernels, 403, 414–417
 - Launchpad, work environments, 638–640
 - LDAP servers, 580
 - local bash shells, 640
 - localhost interfaces, 327
 - MySQL, 494–495
 - networks
 - command line network interfaces, 345–350
 - configuration files, 350–355
 - DHCP, 355–361
 - /etc/host.conf files, 355
 - /etc/hosts files, 350–351
 - /etc/netplan/*.yaml files, 353–355
 - /etc/nsswitch.conf files, 351–352, 355
 - /etc/resolv.conf files, 352–353
 - graphical configuration tools, 355
 - nm-connection-editor, 345–346
 - NFS
 - clients, 423–424
 - servers, 422–423
 - pbuilder, 638
 - PostgreSQL, 498
 - power management, in Ubuntu, 18–19
 - PPP Internet access, 364
 - PPPoE, 364
 - printers, in Ubuntu, 18
 - runtime servers, Apache web server, 446–449
 - Samba, 425
 - /etc/samba/smb.conf files, 426–429
 - global behaviors, 427
 - security, 384–385
 - software repositories, 15–17
 - Squid, 570–572
 - Squid clients, 564–565
 - SSH server, 371–372
 - system settings, in Ubuntu, 17–18
 - Ubuntu
 - date/time settings, 19–20
 - development packages, 637–638, 640–643
 - power management, 18–19
 - printer configurations, 18
 - software repositories, 15–17
 - system settings, 17–18
 - troubleshooting post-configuration problems, 21–22
 - wireless networks, 20

- VPN
 - clients, 573–575
 - servers, 575–577
 - wireless networks, 20
- confining scripts to directories, 198–199
- Conky, 292–297
- connectivity
 - Internet, 363–364
 - checking, 328–330
 - common configuration information, 364–365
 - dial-up access, 367–368
 - DSL access, 365
 - ISP, 364–365
 - Linux commands, 369
 - online references, 369–370
 - PPP access, 364
 - PPPoE, 364
 - troubleshooting, 368–369
 - Ubuntu commands, 369
 - VM, 526
- console-based monitoring, 281–283
- containers, 531
 - Docker, 533
 - Kubernetes, 534
 - LXC, 532
 - LXD, 533
 - online references, 534
- contents of directories, listing, 118–119
- contents of files
 - displaying, 132
 - sorting, 162–163
- converting
 - graphics, 68–69
 - sound files, 65
- copying
 - files between machines (remote access), 372–373

- files/folders, 131, 147, 316–317. *See also* mirroring
 - cp command, 318
 - rsync command, 319–320
 - tar command, 317–318
- VM, 526
- Coreutils, 199
- CouchDB, 515–516
- cp command, 131, 147, 318
- crackers, 380
- createdb command, 506
- createuser command, 506
- cron command, 229–231
- CrossOver Office, 61
- cups cancel command, 435
- cups command, 435
- cups disable command, 435
- cups enable command, 435
- CUPS GUI, network printers, 431–434
- cuspreject command, 435
- custom kernels, 402

D

- D programming language, 624
- daemons
 - AND, 299
 - Linux, 25
 - smbd, starting/stopping, 429
- darktable, 70
- Dart, 624
- dash, 226
- Dash (GNOME), 43–44
- data, SQL tables
 - inserting data, 489–490
 - retrieving data, 490–492

- data directory initialization, PostgreSQL, 498
- data integrity, relational databases, 493–494
- data locking, relational databases, 492
- data loss, 302
- data mirroring, 307
- data recovery, 323
 - GRUB2 boot loaders, restoring, 323–324
 - saving files from nonbooting hard disks, 324
 - Ubuntu Rescue disc, 323
- databases
 - accessing
 - local GUI clients, 503
 - MySQL, 501–503
 - SSH, 501–502
 - web browsers, 503
 - AND operators, 491–492
 - client/server model, 484
 - commands, 506
 - flat file databases, 484–485
 - graphical database clients, 506
 - MySQL, 483
 - ACID compliance, 493–494
 - command line client, 504–505
 - configuring, 494–495
 - creating databases, 496–497
 - data integrity, 493–494
 - data locking, 492
 - database access, 501–503
 - inserting data into tables, 490
 - mysql command, 506
 - mysqladmin command, 506
 - mysqldump command, 506
 - MySQLGUI, 506
 - online references, 493, 507
 - PostgreSQL versus, 492–494
 - procedural languages, 494
 - retrieving data from tables, 490–492
 - root user passwords, 495
 - speed, 492
 - SQL subqueries, 494
 - stored procedures, 494
 - triggers, 494
- NoSQL, 485, 509–510
 - advantages of, 510, 511
 - BaseX, 517
 - Berkeley DB, 512, 579
 - BigTable, 517
 - Cassandra, 513
 - CouchDB, 515–516
 - disadvantages of, 510–511
 - document stores, 514–515
 - etcd, 513
 - FlockDB, 519
 - graph stores, 518
 - HBase, 518
 - HyperGraphDB, 519
 - key/value stores, 512
 - Memcached, 513–514
 - MemcacheDB, 514
 - MongoDB, 516
 - Neo4j, 518
 - NewSQL and, 511
 - online references, 519–520
 - OrientDB, 519
 - Redis, 514
 - Riak, 514
 - Scylla, 514
 - UnQL, 511
 - wide column stores, 517
- OR operators, 491–492
- PostgreSQL
 - ACID compliance, 493–494
 - command line client, 505–506
 - configuring, 498
 - creating database users, 499–500
 - creating databases, 499

- data integrity, 493–494
- data locking, 492
- database access, 501–503
- deleting database users, 500
- granting/revoking privileges, 500–501
- initializing data directories, 498
- MySQL versus, 492–494
- online references, 493, 507
- pgAdmin, 506
- procedural languages, 494
- retrieving data from tables, 491
- speed, 492
- SQL subqueries, 494
- stored procedures, 494
- triggers, 494
- relational databases, 485. *See also* MySQL; PostgreSQL
 - creating tables, 488–489
 - DBA, 483–484
 - inserting data into tables, 489–490
 - operation of, 486–487
 - SQL basics, 487–492
- Datadog, 298
- date/time settings
 - date command, 19
 - Ubuntu, 19–20
- DaVinci Resolve, 77
- DBA, 483–484, 511
- Debian Linux, 24
- debugging C/C++ programming, 602
 - checking source code, 602–603
 - gdb command, 603
 - gprof command, 603
 - splint command, 602–603
 - symbolic debugging, 603
 - tracking function time, 603
- Déjà Dup, 313–314
- deleting
 - directories, 129–131, 161–162
 - PostgreSQL database users, 500
 - user accounts, 137
- deluser command, 137, 209
- dependency checking, 599–600
- depmod command, 407
- desktop environments, 41–42
 - Budgie, 49
 - changing, 39
 - GNOME, 41
 - Dash, 43, 44
 - Mutter, 42
 - Power icon, 43, 45
 - Shell, 43–45
 - Show Applications icon, 43, 44
 - GNOME 2, 48
 - KDE, 45
 - Kubuntu, 45–46
 - Kylin, 50
 - Lubuntu, 47–48
 - LXDE, 47–48
 - MATE, 48–49
 - online references, 50
 - Ubuntu desktops, sharing, 424
 - Ubuntu MATE, 48–49
 - widgets, 42
- X server
 - benefits of, 34
 - components of, 33–34
 - display manager, 39
 - display managers, 33
 - online references, 40
 - starting, 39
 - terminal clients, 33–34
 - window manager, 33–34
 - xorg.conf files, 33–39

- Xfce, 46
- Xubuntu, 46–47
- development (Ubuntu)
 - Bazaar, 639
 - fixing bugs, 640–643
 - GPG keys, 637–639
 - introduction to, 636–637
 - Launchpad, 614–615
 - creating accounts, 638
 - work environments, 638–640
 - local bash shells, configuring, 640
 - online references, 643
 - package installation
 - code packaging, 640–643
 - packages, 637–638, 640–643
 - pbuilder, 638
 - SSH keys, 638, 639
- Device section (xorg.conf files), 37–38
- devices
 - all-in-one (Print/Fax/Scan) devices, 435
 - devices.txt files, 403
 - i2c-dev devices, 407
 - security, 385
- DevOps
 - Chaos Engineering, 614
 - CI/CD tools, 614
 - Sysadmin versus, 535–536
- df command, 286–287
- dhclient command, 369
- DHCP, 338, 361
 - activating, 357
 - dhcpcd, 358
 - Dynamic DNS, 356–357
 - networks
 - configuring, 355–361
 - host configurations, 359–361
 - operation of, 356
 - servers, 358–359
 - software, installing/configuring, 358–359
- dial-up Internet access, configuring, 367–368
- diff command, 170
- differences in files, finding, 170
- digiKam, 70
- digital cameras
 - Ubuntu support, 70
 - USB connections, 70
- directories, 112–114
 - /bin directory, 114
 - changing, 120, 145–147
 - children directories, 118
 - creating, 129, 160
 - data directory initialization, PostgreSQL, 498
 - deleting, 129–131, 161–162
 - DIT, 580
 - /etc directory, 114
 - files
 - copying, 131, 147
 - creating, 128–129
 - displaying contents of, 132
 - finding, 147
 - listing in directories, 156–158
 - moving, 131, 161
 - printing last lines of files, 163–164
 - renaming, 131, 161
 - sorting file contents, 162–163
 - finding current directory, 120
 - home directories, 208, 427–428
 - /home directory, 115
 - kernel directories, 403–405
 - LDAP directories, populating, 582–583
 - listing contents of, 118–119
 - parent directories, 118
 - permissions, 122–127
 - printing directory sizes, 148

- /proc directory, 115–117
- regular expressions, 133
- /sbin directory, 114
- scripts, confining to directories, 198–199
- /tmp directories, 117
- user directories, 115
- /usr directory, 117
 - shared data, 117
 - X server, 33
- /var directory, 117
- wildcard characters, 133
- DirectoryIndex directive (Apache web server), 449
- disaster recovery, planning, 390–391
- disk quotas, 287
 - configuring, 223
 - implementing, 222–223
 - managing, 222
- display managers, X server, 33, 39
- displaying
 - available memory space, 286
 - contents of files, 132
 - free space (hard disks), 286–287
- distributed processing, X server, 32–33
- distributions
 - Linux distributions
 - included software, 24–25
 - versions of, 24
 - Ubuntu distributions, versions of, 27
- DIT, 580
- dmesg command, 21, 200
- DNS, 448–449
 - Dynamic DNS, 356–357
 - online references, 595
 - records, 589–590
 - A records, 590
 - AAAA records, 590
 - CNAME records, 590–591
 - MX records, 591
 - NS records, 591
 - SOA records, 592–593
 - TXT records, 593
 - requests, 589–590
 - search orders, 355
 - servers, 588, 589–590, 593–595
- DocBook, Publican, 59–60
- Docker, 533
- document stores, 514–515
- documentation
 - apropos command, 112
 - games, 89
 - Linux, 28–30
 - manual pages, 111–112
 - mind maps, 61
 - reading, 111–112
 - TeX documents
 - Kile, 61
 - LaTeX, 60
 - LyX, 60
 - Texmaker, 60
 - Ubuntu, 28, 30
 - Wayland, 32
 - whereis command, 112
 - WYGIWYW, 60
- DocumentRoot directive (Apache web server), 449
- domains
 - DNS
 - online references, 595
 - records, 590–593
 - requests, 589–590
 - servers, 588, 589–590, 593–595
 - hostnames, 589
 - names, 588–589
 - root zones, 589
 - TLD, 588–589

- dotdee, 102–103, 322
- Dotfiles, 322
- downloading
 - files/folders, 164
 - Ubuntu updates during installation, 8
- drivers, video drivers (proprietary), installing, 80
- dropdb command, 506
- dropuser command, 506
- DSL Internet access, configuring, 365
- du command, 148
- dual-boot systems, Ubuntu installations, 6
- dumb gateways, 343
- dummy interfaces, 327
- DVD installations, Ubuntu, 2, 6–11
- dvd+rw-tools, 74
- DVD/CD burning applications
 - Brasero, 71
 - Linux command line
 - CD creation, 72–73
 - DVD creation, 73–75
 - preformatted DVD, 74
 - Ubuntu support, 71
- Dynamic DNS, 356–357
- Dynamo, 514

E

- e2fsck command, 397
- echo command, 148–149
- Eclipse, 606
- editing
 - /etc/modprobe.conf files, 344
 - sound, 76
 - Ardour, 76
 - Audacity, 76
 - Cecilia, 76
 - LMMS, 76
 - Mixxx, 76
 - Rosegarden, 76
- text, 185, 186
 - awk, 189–191
 - emacs, 185, 188–189
 - gedit, 186
 - kate, 186
 - kedit, 186
 - nano, 185, 186–187
 - sed, 189–191
 - vi, 185, 187–188
 - vim, 185
- video, 76, 77
 - Avidemux, 77
 - Blender, 77
 - Cinelerra, 77
 - DaVinci Resolve, 77
 - Kdenlive, 77
 - Lightworks, 77
 - OpenShot Video Editor, 77
 - PiTiVi, 77
 - Shotcut, 77
- Elixir, 625
- Elm, 625
- emacs text editor, 185, 188–189
- email, 547
 - Autoresponders, 562
 - virus scanners, 562
- embedded spaces, resolving strings with, 248–249
- emulators, 79
- encrypted files, 318
- endless loops, 261–262
- enterprise servers, monitoring, 298–299
- environment variables, 182–185, 242
- erasing, hard disks, Ubuntu installations, 9
- Erlang, 625

- error messages
 - redirecting input/output of commands, 169–170
 - sudo command, 134–136
- /etc directory, 114
- etcd, 513
- /etc/host.conf files, 355
- /etc/hosts files, 350–351
- /etc/modprobe.conf files, 344, 407
- /etc/modprobed files, 407
- /etc/netplan/*.yaml files, 353–355
- /etc/nsswitch.conf files, 351–352, 355
- /etc/resolv.conf files, 352–353
- /etc/samba/smb.conf files, 426–429
- etckeeper, 321–322
- ethereal command, 369
- Ethernet
 - 10G Ethernet networking, 340
 - 50G Ethernet networking, 340
 - Gigabit Ethernet. See 1000BASE-T networking
 - PPPoE, 364
- evaluating backup strategies, 304–306
- event handlers, 468–469
- executing
 - jobs in parallel, 181–182
 - shell scripts, 237–239
- exit statements, 269
- expr statements, 266
- expressions, 235
 - comparisons of expressions, 250–251
 - file operators, 253–254, 257–258
 - logical operators, 255, 259
 - numeric comparisons, 252–253, 256–257
 - strings, 251–252, 255–256
 - directories, 133
- external attacks, 379–380
- external hard drives, backups, 308

F

- faillog command, 289
- fax machines, all-in-one (Print/Fax/Scan) devices, 435
- FDDI networks, 340
- Fetchmail, 557–560
- fg command, 174–175
- fiber-optic networks, 340, 342
- File Roller, 312
- file systems
 - authentication, Apache web server, 452
 - directories, 112–114
 - /bin directory, 114
 - changing, 120, 145–147
 - children directories, 118
 - copying files, 131, 147
 - creating, 129, 160
 - creating files, 128–129
 - deleting, 129–131, 161–162
 - displaying contents of files, 132
 - /etc directory, 114
 - finding current directory, 120
 - finding files, 147
 - /home directory, 115
 - listing contents of, 118–119
 - listing files in directories, 156–158
 - moving files, 131, 161
 - parent directories, 118
 - printing directory sizes, 148
 - printing last lines of files, 163–164
 - /proc directory, 115–117
 - regular expressions, 133
 - renaming files, 131, 161
 - /sbin directory, 114
 - sorting file contents, 162–163
 - /tmp directories, 117
 - /usr directory, 117
 - /var directory, 117

- wildcard characters, 133
- hierarchy of, 112–114
- permissions, 120–121
 - altering, 123–124
 - assigning, 121–122
 - changing file groups, 125
 - changing file permissions, 125, 147
 - setting with ACL, 127–128
 - sgid permissions, 125–127
 - sticky bit permissions, 126–127
 - suid permissions, 125–127
 - viewing/modifying default permissions, 124–125
- Files section (xorg.conf files), 35–36
- files/folders
 - apache2.conf files, 447–449
 - comparisons, 170
 - compressed files, 191–192, 318
 - configuration files, version control, 320–322
 - copying. *See also* mirroring, 131–132, 147, 316–317
 - between machines (remote access), 372–373
 - cp command, 318
 - rsync command, 319–320
 - tar command, 317–318
- creating in directories, 128–129
- disabling access time, 397
- displaying contents of, 132
- Dotfiles, 322
- downloading, 164
- encrypted files, 318
- /etc/host.conf files, 355
- /etc/hosts files, 350–351
- /etc/modprobe.conf files, 407
- /etc/modprobed files, 407
- /etc/netplan/*.yaml files, 353–355
- /etc/nsswitch.conf files, 351–352, 355
- /etc/resolv.conf files, 352–353
- file operator comparisons, 253–254, 257–258
- File Roller, 312
- finding, 149–151
 - differences in files, 170
 - from indexes, 156
 - similarities in files, 170–171
 - in Ubuntu, 12
- inodes, 155
- LDIF files, 583
- links, creating, 154–156
- listing in directories, 156–158
- log files
 - checking, 287–289
 - rotating, 289–291
- makefiles, 599–600
- mirroring. *See also* copying, 164
- moving, 131, 161
- network configuration files, 350–355
- new user home directories, 208
- NFS, 422
 - clients, 423–424
 - installing, 422
 - servers, 422–423
 - starting/stopping, 422
- password files, 212–214
- permissions, 204
 - altering, 123–124
 - changing file groups, 125
 - changing permissions, 125, 147
 - viewing/modifying default permissions, 124–125
- printing
 - contents of, 144–145
 - file sizes, 148
 - last lines of files, 163–164
- renaming, 131, 161
- restoring files from archives, 311–312

- Samba, 424
 - configuring, 425, 427–429
 - /etc/samba/smb.conf files, 426–429
 - installing, 425–426
 - saving files from nonbooting hard disks, 324
 - sharing
 - NFS, 422–424
 - online references, 435
 - Samba, 424–431
 - sorting contents of, 162–163
 - temporary file storage, /tmp directories, 117
 - time warps, 212
 - variable data files, /var directory, 117
 - filesystems
 - performance tuning, 394–395
 - synchronizing, 396
 - find command, 126
 - finding
 - directories, 120
 - files/folders, 149–151
 - differences in files, 170
 - from indexes, 156
 - similarities in files, 170–171
 - in Ubuntu, 12
 - groups, 206
 - software, with APT, 98–99
 - finger information fields, 213
 - Firefox, 52–53
 - firewalls
 - configuring, 386–388
 - iptables, 388
 - nftables, 388
 - UFW, 386–387
 - .flac files, 65
 - flat file databases, 484–485
 - flexbackup, 316
 - FlightGear, 87
 - FlockDB, 519
 - .flv files, 75
 - folders. *See* files/folders
 - for statements, 259–261
 - foreground jobs, 174–175
 - formatting, DVD, 74
 - Forth, 626
 - Fortran, 626
 - FQDN, 448–449, 580
 - free command, 286
 - free/open-source software, Office compatibility, 55
 - free space, displaying, 286–287
 - hard disks, 286–287
 - memory space, 286
 - Frets on Fire, 86
 - Frozen Bubble, 84
 - fstab command, 114
 - ftp command, 372
 - full backups
 - with incremental backups, 307
 - on a periodic basis, 306–307
 - tar command, 310–311
 - functions, shell scripts, 269–270
- ## G
- Game Jolt, 82
 - games, 79, 80
 - Battle for Wesnoth, 85–86
 - commercial games, 88
 - documentation, 89
 - emulators, 79
 - FlightGear, 87
 - Frets on Fire, 86
 - Frozen Bubble, 84

- Game Jolt, 82
- GOG.com, 82
- Humble, 82
- installing
 - from Ubuntu repositories, 82–87
 - video drivers (proprietary), 80
- itch.io, 82
- kid-friendly games, 88
- LGDB, 82
- online game sources, 81–82
- online references, 89
- Scorched 3D, 83–84
- Speed Dreams, 87
- Steam, 81
- SuperTux, 84–85
- video drivers (proprietary), installing, 80
- Warsow, 82
- Windows games, 88–89
- gateways
 - dumb gateways, 343
 - smart gateways, 343
- GCC, 597, 603–604
- gdb command, 603
- gedit, 59
- gedit text editor, 186
- genprof, 389
- GHC, 627
- GID, 204, 448
- .gif files, 68
- Gigabit Ethernet. *See* 1000BASE-T networking
- GIMP, 66–67
- Git, software project management, 610–611
- Glade, 607
- GNOME, 41
 - Dash, 43, 44
 - File Roller, 312
 - Glade, 607
 - Mutter, 42
 - performance tuning, 3933
 - Power icon, 43, 45
 - Shell, 43–45
 - Show Applications icon, 43, 44
- GNOME 2, 48
- gnome-nettool, 297
- GNU
 - Coreutils, 199
 - GCC, 597, 603–604
 - GPL, development of, 23–24
- Go programming language, 626–627
- GOG.com, 82
- Google Chrome, 53–54
- Google Chromium, 53–54
- Google Cloud, 539
- Google Play, 607
- GParted, 9–10
- gpsswd command, 207
- GPG keys
 - configuring, 637–638
 - uploading, 638–639
- GPL, development of, 23–24
- gprof command, 603
- GPT, boot process, 274
- granting/revoking privileges, PostgreSQL, 500–501
- graph stores, 518
- graphical database clients, 506
- graphical development tools, 605
 - Eclipse, 606
 - Glade, 607
 - IDE, 605–606
 - KDevelop Client, 606–607
 - NetBeans, 606
 - Oracle JDeveloper, 606
 - SDK, 606
 - Visual Studio Code, 606

- graphical network configuration tools, 355
- graphical process tools, 292
 - Conky, 292–297
 - gnome-nettool, 297
 - vncviewer, 297
 - Wireshark, 298
- graphics, 67–68
 - .bmp files, 68
 - converting, 68–69
 - .gif files, 68
 - image captures, 69
 - .jpg files, 68
 - .pbm files, 69
 - .pcx files, 68
 - .pgm files, 69
 - .png files, 68
 - .pnm files, 69
 - .ppm files, 69
 - .svg files, 68
 - .tif files, 68
- graphics manipulation applications, 66, 70
 - Blender, 70, 77
 - CinePaint, 70
 - darktable, 70
 - digiKam, 70
 - GIMP, 66–67
 - Hugin, 70
 - Inkscape, 70
 - Krita, 70
 - nautilus-image converter, 69
 - netpbm tools, 69
 - Photoshop, 67
 - POV-Ray, 70
 - Radiance, 70
 - Shotwell Photo Manager, 66, 71
 - Simple Scan, 67
 - Xara Xtreme, 70
- grep command, 151–152, 234
- Groovy, 627
- Group directive (Apache web server), 448
- group management, 205
 - finding groups, 206
 - listing groups, 205–206
 - tools, 206–207
- groupadd command, 207
- groupdel command, 207
- groupmod command, 207
- groups command, 224
- grpck command, 207
- GRUB
 - recovery mode, 196
 - reinstalling, 195–196
 - Ubuntu bootups, troubleshooting, 195–196
- GRUB2
 - boot loaders, restoring, 323–324
 - installing, 5
- Guacamole, 377
- GUI, 31, 41–42
 - Budgie, 49
 - GNOME, 41
 - Dash, 43, 44
 - Mutter, 42
 - Power icon, 43, 45
 - Shell, 43–45
 - Show Applications icon, 43, 44
 - GNOME 2, 48
 - KDE, 45
 - Kubuntu, 45–46
 - Kylin, 50
 - local GUI clients, database access, 503
 - Lubuntu, 47–48
 - LXDE, 47–48
 - MATE, 48–49
 - MySQLGUI, 506

- pgAdmin, 506
 - Ubuntu MATE, 48–49
 - Wayland, 32
 - widgets, 42
 - X server, 32, 33
 - benefits of, 33–34
 - components of, 33–34
 - display manager, 39
 - display managers, 33
 - distributed processing, 32–33
 - online references, 40
 - starting, 39
 - terminal clients, 33–34
 - /usr directory, 33
 - window manager, 33–34
 - xorg.conf files, 33–39
 - Xfce, 46
 - Xubuntu, 46–47
 - gunzip command, 191
 - gzip command, 191
- ## H
- hackers, 380
 - hard disks
 - disk quotas, 287
 - erasing, Ubuntu installations, 9
 - free space, displaying, 286–287
 - nonbooting disks, saving files from, 324
 - printing, disk usage, 148
 - Ubuntu installations, erasing hard disks, 9
 - hard drives (external), backups, 308
 - hard links, 155–156
 - hardware
 - Mac hardware, Ubuntu compatibility, 3
 - network hardware, initializing, 343–344
 - /etc/modprobe.conf files, 343–344
 - kernel modules, 344–345
 - modprobe command, 344–345
 - Ubuntu specifications
 - Mac hardware, 3
 - researching, 2–16
 - Haskell, 627
 - HBase, 518
 - hdparm command, 395–396
 - Heimer, 61
 - help, Ubuntu online references, 22
 - history of commands, viewing, 197
 - hitsujiTMO, 272
 - home applications, Ubuntu for, 28
 - home directories, 208
 - /home directory, 115
 - home users, backup strategies, 304–305
 - Horowitz, Eliot, 516
 - hosting (virtual), Apache web server, 461–462
 - hostnames, 589
 - hosts (networks)
 - adding to /etc/hosts files, 350–351
 - configuring with DHCP, 359–361
 - Nginx web server, 473–474
 - HOWTO documents, Linux, 29
 - HPC, 598
 - .htaccess configuration files, Apache web server, 450, 469
 - AllowOverrides directives, 451–452
 - Options directives, 451
 - HTTP servers
 - Caddy, 482
 - Cherokee, 480–481
 - Jetty, 481
 - lighttpd, 479–480
 - online references, 482
 - thttpd, 481–482

- Tomcat, 482
 - Wildfly, 482
 - Yaws, 480
 - HTTPS
 - Apache web server, 464–466
 - Nginx web server, 476–477
 - hubs, 342
 - Hugin, 70
 - Humble, 82
 - hwclock command, 19–20
 - hybrid clouds, 539–540
 - HyperGraphDB, 519
- I
- i2c-dev devices, 407
 - laaS, 537
 - ID
 - GID, 204, 448
 - PID, 164, 175–176, 237, 274, 566
 - sgid permissions, 125–127
 - suid permissions, 125–127
 - UID, 204, 448
 - IDE, 605–606
 - ide.txt files, 404
 - if statements, 265–266
 - ifconfig command, 346–348, 369, 385
 - image captures, 69
 - image file formats, 67–68
 - .bmp files, 68
 - .gif files, 68
 - .jpg files, 68
 - .pbm files, 69
 - .pcx files, 68
 - .pgm files, 69
 - .png files, 68
 - .pnm files, 69
 - .ppm files, 69
 - .svg files, 68
 - .tif files, 68
 - image scanners, 67
 - incremental backups
 - full backups with incremental backups, 307
 - tar command, 310–311
 - indexes, finding files from, 156
 - init systems, 272
 - initializing
 - data directories, PostgreSQL, 498
 - network hardware, 343–344
 - /etc/modprobe.conf files, 344
 - kernel modules, 344–345
 - modprobe command, 344–345
 - initrd.txt files, 404
 - Inkscape, 70
 - inodes, 155
 - InputDevice section (xorg.conf files), 36–37
 - input/output of commands
 - redirecting, 167–170
 - redirecting with shell command line, 236
 - standard input/output, 167, 169
 - input strings, searches, 151–152
 - inserting data into SQL tables, 489–490
 - insmod command, 406
 - installing
 - Android Studio, 609
 - Apache web server, 444
 - DHCP activation at installation, 357
 - DHCP software, 358–359
 - games, from Ubuntu repositories, 82–87
 - GRUB, 195–196
 - GRUB2, 5
 - NFS, 422
 - Nginx web server, 469–470

- Samba, 425–426
 - Squid, 564
 - Ubuntu, 196
 - 32-bit Ubuntu, 4–5
 - 64-bit Ubuntu, 4–5
 - backups, 7
 - boot loaders, 5–6
 - cloud computing, 536
 - downloading updates during installation, 8
 - dual-boot systems, 6
 - DVD installations, 2, 6–11
 - erasing hard disks during installation, 9
 - flavors of Ubuntu, 2–3
 - language selection, 7–8
 - Mac hardware compatibility, 3
 - partition strategies, 5
 - partitions, 5, 9
 - password creation, 10
 - preinstallation process, 1–2
 - researching hardware specifications, 2–16
 - storage drives, 9
 - UEFI, 6
 - USB thumb drive installations, 3, 6–11
 - Ubuntu development packages
 - code packaging, 640–643
 - packages, 637–638, 640–643
 - video drivers (proprietary), 80
 - virt-install, 525–526
- instances, 533
- internal attacks, 379–380
- Internet, 51–52
- Chrome, 53–54
 - Chromium, 53–54
 - connections, 363–364
 - checking, 328–330
 - common configuration information, 364–365
 - dial-up access, 367–368
 - DSL access, 365
 - ISP, 364–365
 - Linux commands, 369
 - online references, 369–370
 - PPP access, 364
 - PPPoE, 364
 - troubleshooting, 368–369
 - Ubuntu commands, 369
 - dummy interfaces, 327
 - Firefox, 52–53
 - introduction to, 51
 - ISP, 364–365
 - localhost interfaces, 326
 - checking availability of, 326
 - configuring manually, 327
 - I/O transfer speeds, 396
 - IP addressing
 - IPv4 addressing, 331–332
 - IPv6 addressing, 332, 334–336
 - Squid, specifying client IP addresses, 569–570
 - ip command, 348–349
 - IP masquerading, 332–333
 - ip route command, 346, 349
 - iptables, 388
 - ISP, 364–365
 - itch.io, 82
 - iwconfig command, 369
- ## J
- Java programming, 604, 627–628
 - JVM, 604
 - OpenJDK, 604–605
 - JavaScript, 628
 - JDeveloper (Oracle), 606

Jetty, 481

jobs

- background jobs, 174–175
- foreground jobs, 174–175
- listing, 173
- moving, to background/foreground, 174–175
- parallel, executing in, 181–182
- running repeatedly, 229–231

jobs command, 173

.jpg files, 68

JSON, 628

Juju, 540–541, 545

JVM, 604

K

kate text editor, 186

kdat, 312–313

KDE, 45

- kdat, 312–313
- KDE ark archiving tool, 312–313
- monitoring tools, 298
- performance tuning, 3933

Kdenlive, 77

KDevelop Client, 606–607

kdf, 298

kedit text editor, 186

kernel ring buffers, reading contents of, 200

kernels, 401, 402

- Android mobile development, 608
- compiling, 418–419
- configuring, 403, 414–417
- custom kernels, 402
- devices.txt files, 403
- directories, 403–405
- first kernel, 402
- ide.txt files, 404
- initrd.txt files, 404
- kernel oops, 419
- kernel-parameters.txt files, 404
- KVM, 523–526
- loading, 274
- loading process, 344–345
- make utility, 404
- modules, 406–408
- numbering schema, 24
- obtaining sources, 409–410
- online references, 419–420
- patching, 410–411
- performance tuning, 398–399
- /proc directory, 115–117
- RAM disk images, creating, 418
- recompiling, 408–409
- security, 379
- source tree, 403–405
- sysrq.txt files, 404
- troubleshooting, 418–419
- types of, 405–406
- Ubuntu kernels, 401–402
- versions of, 409

key-based logins, 373–375

key/value stores, 512

kid-friendly games, 88

Kile, 61

kill command, 283–284

killall command, 284

Kotlin, 628–629

Krita, 70

ksh, 226

ksysguard, 298

Kubernetes, 534

Kubuntu, 45–46

KVM, 523–526

Kylin, 50

L

LAMP, 437–439

LAN, network printers, 431–432

Landscape, 541, 545

languages

procedural languages, relational databases, 494

programming languages, 621–622

Ada, 622

Clojure, 622–623

COBOL, 623–624

D, 624

Dart, 624

Elixir, 625

Elm, 625

Erlang, 625

Forth, 626

Fortran, 626

Go, 626–627

Groovy, 627

Haskell, 627

Java, 627–628

JavaScript, 628

Kotlin, 628–629

Lisp, 629

Lua, 629

Mono, 629–630

OCaml, 630

online references, 633–634

Perl, 630

PHP, 631

Python, 631

Raku, 631

Ruby, 631–632

Rust, 632

Scala, 632

Scratch, 632–633

Vala, 633

selecting, Ubuntu installations, 7–8

large enterprises, backup strategies, 305

last command, 212

last lines of files, printing, 163–164

lastlog command, 289

later, scheduling tasks for, 227–229

LaTeX, 60

Launchpad, 614–615

accounts, creating, 638

GPG keys, 638–639

SSH keys, 639

work environments, configuring, 638–640

LDAP, 579–580

administration, 584–585

configuring

clients, 584

servers, 580

directories, populating, 582–583

ldap-utils package, 584–585

LDIF files, 583

online references, 585

schemas, creating, 580–581

Thunderbird, LDAP client configuration, 584

LDIF files, 583

LEMP, 439

less command, 132, 152–154

LGDB, 82

libraries (software), Android mobile development, 608

LibreOffice

brief history of, 57–58

components of, 56–57

Office compatibility, 55

VBA, 55

lighttpd, 479–480

Lightworks, 77

links

files/folders, creating links, 154–156

hard links, 155–156

symlinks, 154–156

Linux

C programming, 598–599

command line

&& operators, 180, 198

!! operator, 196–197

|| operator, 198

accessing, 107

apropos command, 112

background jobs, 174–175

basic commands, 143–144

BIOS, 195

boolean operators, 180

byobu, 192–193

CD creation, 72–73

changing directories, 145–147

command shortcuts, 198

commands list, 143–144

commonly used commands/programs,
139

compressed files, 191–192

confining scripts to directories, 198–199

copying files, 147

Coreutils, 199

creating links, 154–156

defined, 106–107

directories, 112–117

downloading files/folders, 164

DVD creation, 73–75

environment variables, 182–185

executing jobs in parallel, 181–182

file comparisons, 170

file systems, 112–114

finding differences in files, 170

finding files, 149–151

finding files from indexes, 156

finding similarities in files, 170–171

foreground jobs, 174–175

GRUB, 195–196

input string searches, 151–152

kernel ring buffers, 200

listing files in directories, 156–158

listing jobs, 173

listing processes, 171–173

listing system information, 158–159

logging out of, 108

manual pages, 111–112, 159–160

multiple terminal operation, 192–193

online references, 139, 165, 200

pipng commands, 178–180

polite system resets, 194–195

printing directory sizes, 148

printing disk usage, 148

printing file contents, 144–145

printing resource usage, 175–177

prioritizing processes, 177–178

reading documentation, 111–112

reasons for using, 142–143

rebooting systems, 138–139

redirecting input/output of commands,
167–170

REISUB, 194–195

remote computer logins, 108–109

repeating text, 148–149

rerunning previous commands, 196–197

root accounts, 110–111, 133–134

running background processes, 173–174

running commands in sequence,
180–181

shutting down systems, 137–138

- standard input/output of commands, 169
 - substituting processes, 181
 - super user accounts, 109–111, 133–134
 - text editors, 185–191
 - text-based logins, 107–108
 - troubleshooting sudo command, 134–136
 - troubleshooting Ubuntu bootups, 195
 - user accounts, 109–111, 136–137
 - viewing command history, 197
 - whereis command, 112
- daemons, 25
- Debian Linux, 24
- development of, 23–25
- directories, 112–114
- /bin directory, 114
 - changing, 120, 145–147
 - children directories, 118
 - copying files, 131, 147
 - creating, 129, 160
 - creating files, 128–129
 - deleting, 129–131, 161–162
 - displaying contents of files, 132
 - /etc directory, 114
 - finding current directory, 120
 - finding files, 147
 - /home directory, 115
 - listing contents of, 118–119
 - listing files in directories, 156–158
 - moving files, 131, 161
 - parent directories, 118
 - permissions, 122–127
 - printing directory sizes, 148
 - printing last lines of files, 163–164
 - /proc directory, 115–117
 - regular expressions, 133
 - renaming files, 131, 161
 - /sbin directory, 114
 - sorting file contents, 162–163
 - /tmp directories, 117
 - /usr directory, 117
 - /var directory, 117
 - wildcard characters, 133
- distributions
- included software, 24–25
 - versions of, 24
- documentation, 28–30
- file systems
- directories, 112–120
 - hierarchy of, 112–114
 - permissions, 120–128
- filesystems
- performance tuning, 394–395
 - synchronizing, 396
- GNU GPL, development of, 23–24
- HOWTO documents, 29
- kernels, 401, 402
- Android mobile development, 608
 - compiling, 418–419
 - configuring, 403, 414–417
 - creating RAM disk images, 418
 - custom kernels, 402
 - devices.txt files, 403
 - directories, 403–405
 - first kernel, 402
 - ide.txt files, 404
 - initrd.txt files, 404
 - kernel oops, 419
 - kernel-parameters.txt files, 404
 - loading, 274
 - make utility, 404
 - modules, 406–408
 - obtaining sources, 409–410
 - online references, 419–420
 - patching, 410–411
 - performance tuning, 398–399

- recompiling, 408–409
- security, 379
- source tree, 403–405
- sysrq.txt files, 404
- troubleshooting, 418–419
- types of, 405–406
- versions of, 409
- location of commands, printing, 164
- manual pages, 29–30
- network connectivity commands, 369
- numbering schema, 24
- online references, 23
- paging through output, 152–154
- reasons for using, 25–26
- rebooting systems, 138–139
- security, 385
- SELinux, 388
- shells, 25
- shutting down systems, 137–138
- user accounts
 - creating, 136–137
 - deleting, 137
- video, viewing, 76
- viruses, 385
- web resources, 23
- X server, 32, 33
 - benefits of, 33–34
 - components of, 33–34
 - display manager, 39
 - display managers, 33
 - distributed processing, 32–33
 - online references, 40
 - starting, 39
 - terminal clients, 33–34
 - /usr directory, 33
 - window manager, 33–34
 - xorg.conf files, 33–39
- Lisp, 629
- Listen directive (Apache web server), 447
- listing
 - contents of directories, 118–119
 - files/folders in directories, 156–158
 - groups, 205–206
 - jobs, 173
 - processes, 171–173
 - system information, 158–159
 - units (services), 276
- LMMS, 76
- In command, 154–156
- loading process
 - kernels, 274, 344–345
 - modprobe command, 344–345
 - modules, 408
- local bash shells, configuring, 640
- local GUI clients, database access, 503
- localhost interfaces, 326
 - checking availability of, 326
 - configuring manually, 327
- locate command, 156
- location of commands, printing, 164
- locking
 - data, relational databases, 492
 - users out of accounts, 209
- log files
 - checking, 287–289
 - rotating, 289–291
- logging, Apache web server, 463–464
- logging out of command line, 108
- logical operators, comparisons, 255, 259
- logins
 - command line
 - from remote computers, 108–109
 - text-based logins, 107–108
 - key-based logins, 373–375
- logname command, 224

loopback interfaces. See localhost interfaces

loops (endless), 261–262

loss of data, 302

lp command, 435

lpc command, 435

lpq command, 435

lprm command, 435

lpstat command, 435

ls command, 118–119, 156–158

lsblk command, 158–159

lshw command, 158–159

lsmod command, 158–159, 406–408

lspci command, 158–159

Lua, 629

Lubuntu, 47–48

users, 205

LXC, 532

LXD, 533

LXDE, 47–48

LyX, 60

M

MaaS, 537

Mac hardware, Ubuntu installations, 3

MAC systems, 388

macros, 600–601

Mail Delivery Agent (MDA), 560–562

Mail Transport Agent (MTA), 548

Maildir Versus Mbox, 550

make utility, 404

makefiles, 599–600

managing

 Apache web server

 .htaccess configuration files, 450–452, 469

 access control, 452, 455

 configuring, 446–449, 450–452

 development of, 443–444

 file system authentication, 45–455

 HTTPS, 464–466

 installing, 444

 logging, 463–464

 modules, 455–461

 MPM, 449–450

 online references, 466

 overview of, 444

 runtime servers, 446–449

 security, 444, 452–455

 starting/stopping, 444–446

 threads, 468

 virtual hosting, 461–462

 configurations, 102

 dotdee, 102–103

 Snappy package manager, 103–104

 Ubuntu Core, 103

 disk quotas, 222

 groups, 205

 finding groups, 206

 listing groups, 205–206

 tools, 206–207

 modules, 406–408

 Nginx web server, 467–469

 building sources, 469–470

 configuring, 470–473

 HTTPS, 476–477

 installing, 469–470

 modules, 475

 online references, 477

 PHP, 474–475

 uninstalling, 470

 virtual hosting, 473–474

 passwords, 212, 216

 changing in batches, 216–217

 password files, 212–214

 shadow passwords, 214–215

 system password policies, 212

- sets of servers, 543
 - Ansible, 544
 - CFEngine, 545
 - Chef, 544
 - Juju, 545
 - Landscape, 545
 - online references, 545
 - Puppet, 543–544
 - SaltStack, 544–545
- software
 - APT, 95–100
 - apt-get, 99–100
 - compiling software, 100–102
 - configuration management, 102–104
 - online references, 104
 - Software Updater, 94–95
 - Synaptic, 92–94
 - Ubuntu Software, 91
- software project management
 - Bazaar, 611–612
 - Git, 610–611
 - Subversion, 612–613
- system management tools, 292
 - Conky, 292–297
 - gnome-nettool, 297
 - System Monitor, 292
 - vncviewer, 297
 - Wireshark, 298
- system resource management
 - APT, 95–100
 - apt-get, 99–100
 - compiling software, 100–102
 - configuration management, 102–104
 - online references, 104
 - Software Updater, 94–95
 - Synaptic, 92–94
 - Ubuntu Software, 91
- user accounts, 201–202, 204, 207
 - adding accounts, 209–211
 - administration privileges, 217
 - BOFH, 205
 - changing identities, 217–219
 - disk quotas, 222–223
 - file permissions, 204
 - GID, 204
 - group management, 205–207
 - locking users out of accounts, 209
 - lusers, 205
 - monitoring user activity, 211–212
 - online references, 224
 - passwords, 202, 212–217
 - root privileges, 219–222
 - root user accounts, 202–203
 - stereotypes, 205
 - super user accounts, 202, 203
 - system user accounts, 203–204
 - tools, 208–209
 - Ubuntu commands, 223–224
 - UID, 204
 - usernames, 211
- manual pages, 111–112
 - Linux, 29–30
 - reading, 159–160
- MATE, 48–49
- MBR, boot process, 274
- MEAN, 440–441
- Memcached, 513–514
- MemcacheDB, 514
- memory, displaying available space, 286
- Merriman, Dwight, 509
- message block printing, 432–433
- microservice architectures, 531
- Microsoft Azure, 539
- mind maps, 61

- mirroring data, 307
- mirroring. *See also* copying, 164
- Mixxx, 76
- mkdir command, 129, 160
- mkisofs command, 72
- "moat mentality", 333
- mobile development, Android, 607
 - Android Runtime, 608
 - Android Studio, 609
 - Application Framework, 608–609
 - applications, 609
 - ARM processors, 608
 - Google Play, 607
 - Linux kernels, 608
 - RISC processors, 608
- modifying default file permissions, 124–125
- modinfo command, 407
- modprobe command, 344–345, 406–407
- modprobe.d/ command, 114
- Module section (xorg.conf files), 36
- modules, 406–408
 - Apache web server, 455–461
 - MPM, Apache web server, 449–450
 - Nginx web server, 475
- Mojo, 541
- MongoDB, 516
- Monitor section (xorg.conf files), 37
- monitoring
 - console-based monitoring, 281–283
 - enterprise servers, 298–299
 - KDE monitoring tools, 298
 - systems
 - checking log files, 287–289
 - Conky, 292–297
 - console-based monitoring, 281–283
 - disk quotas, 287
 - displaying free hard disk space, 286
 - displaying used memory, 286
 - enterprise servers, 298–299
 - graphical process tools, 292–298
 - KDE monitoring tools, 298
 - killing processes, 283–284
 - online references, 299
 - priority scheduling, 285
 - rotating log files, 289–291
 - system management tools, 292–298
 - System Monitor, 292
 - user activity, 211–212
- Mono, 629–630
- motd files, 138
- mounting, Samba shares, 430–431
- .mov files, 75
- moving
 - files/folders, 131, 161
 - jobs
 - background jobs, 174–175
 - to background/foreground, 174–175
 - foreground jobs, 174–175
- .mp3 files, 65
- .mp4 files, 65
- .mpeg files, 75
- MPM, Apache web server, 449–450
- MS Office
 - CrossOver Office, 61
 - free/open-source software compatibility, 55
 - LibreOffice compatibility, 55
 - VBA, 55
- MS-DOS, 107
- mtr command, Internet connections, checking, 329–330
- multicasting, 338
- multimedia applications
 - ALSA, 64
 - CD/DVD burning applications

- Brasero, 71
- Linux command line, CD creation, 72–73
- Linux command line, DVD creation, 73–75
- Ubuntu support, 71
- graphics manipulation applications, 66, 70
 - Blender, 70, 77
 - CinePaint, 70
 - darktable, 70
 - digiKam, 70
 - GIMP 66–67
 - Hugin, 70
 - Inkscape, 70
 - Krita, 70
 - nautilus-image converter, 69
 - netpbm tools, 69
 - Photoshop, 67
 - POV-Ray, 70
 - Radiance, 70
 - Shotwell Photo Manager, 66, 71
 - Simple Scan, 67
 - Xara Xtreme, 70
- music applications, 65
 - Banshee, 66
 - Rhythmbox, 66
 - Sound Juicer, 66
- online references, 77
- OSS, 64
- PulseAudio, 64
- recording/editing sound, 76
- sound cards, 63–64
- video editing applications, 77
 - Avidemux, 77
 - Blender, 77
 - Cinelerra, 77
 - DaVinci Resolve, 77
 - Kdenlive, 77
 - Lightworks, 77
 - OpenShot Video Editor, 77
 - PiTiVi, 77
 - Shotcut, 77
- multiple jobs, executing in parallel, 181–182
- multiple terminal operation, byobu, 192–193
- music applications, 65
 - Banshee, 66
 - Rhythmbox, 66
 - Sound Juicer, 66
- Mutter, 42
- mv command, 131, 161
- MX records, 591
- MySQL, 483
 - ACID compliance, 493–494
 - command line client, 504–505
 - configuring, 494–495
 - creating databases, 496–497
 - data integrity, 493–494
 - data locking, 492–493
 - database access, 501
 - local GUI clients, 503
 - SSH, 501–502
 - web browsers, 503
 - inserting data into tables, 490
 - mysql command, 506
 - mysqladmin command, 506
 - mysqldump command, 506
 - MySQLGUI, 506
 - online references, 493, 507
 - PostgreSQL versus, 492–494
 - procedural languages, 494
 - retrieving data from tables, 490–492
 - root user passwords, 495
 - speed, 492
 - SQL subqueries, 494
 - stored procedures, 494
 - triggers, 494

mysql command, 506
 mysqladmin command, 506
 mysqldump command, 506
 MySQLGUI, 506

N

Nagios, 298

name servers

DNS

online references, 595

records, 590–593

requests, 589–590

servers, 588, 589–590, 593–595

setting, 352–355

naming services, 351–352

nano text editor, 185, 186–187

NAS, 308

NAT, 332

nautilus-image converter, 69

needs assessments, backup strategies,
 303–304

Neo4j, 518

neofetch command, 158–159

NetBeans, 606

NetBoot, boot process, 274

NetCat, 330

netmasks. *See* subnet masks

netpbm tools, 68–69

netstat command, 346, 350

networks, 337

10BASE-T networking, 339

10G Ethernet networking, 340

50G Ethernet networking, 340

100BASE-T networking, 339

1000BASE-T networking, 339

accessing, chains, 387

bridges, 343, 524–525

broadcasting, 338

cabling, 340

fiber-optic cabling, 342

UTP cabling, 341

chains, 387

command line network interface configuration

ifconfig command, 346–348

ip command, 348–349

ip route command, 346, 349

netstat command, 346, 350

configuring

command line network interfaces,
 345–350

configuration files, 350–355

DHCP, 355–361

/etc/host.conf files, 355

/etc/hosts files, 350–351

/etc/netplan/*.yaml files, 353–355

/etc/nsswitch.conf files, 351–352, 355

/etc/resolv.conf files, 352–353

graphical configuration tools, 355

nm-connection-editor, 345–346

connections, troubleshooting, 342

DHCP, 361

activating, 357

configuring hosts, 359–361

installing/configuring software, 358–359

DNS search orders, 355

dummy interfaces, 327

FDDI networks, 340

fiber-optic networks, 340, 342

gateways

dumb gateways, 343

smart gateways, 343

hosts

adding to /etc/hosts files, 350–351

configuring with DHCP, 359–361

- hubs, 342
- initializing hardware, 343–344
 - /etc/modprobe.conf files, 344
 - kernel modules, 344–345
 - modprobe command, 344–345
- Internet connections, 363–364
 - checking, 328–330
 - common configuration information, 364–365
 - dial-up access, 367–368
 - DSL access, 365
 - ISP, 364–365
 - Linux commands, 369
 - PPP access, 364
 - PPPoE, 364
 - troubleshooting, 342
 - Ubuntu commands, 369
- IP masquerading, 332–333
- IPv4 addressing, 331–332
- IPv6 addressing, 332, 334–336
- LAN, network printers, 431–432
- localhost interfaces, 326
 - checking availability of, 326
 - configuring manually, 327
- "moat mentality", 333
- multicasting, 338
- name servers, 352–355
- naming services, 351–352
- NAT, 332
- NIC, 338–340
- online references, 369–370
- printers, 431
 - CUPS GUI, 431–434
 - LAN, 431–432
 - online references, 435
 - server message block printing, 432–433
 - troubleshooting, 434–435
- routers, 343
- storage, backups, 308
- subnet masks, 337–338
- subnetting, 337–338
- switches, 342
- TCP/IP, 330
 - addressing, 331–332
 - IP masquerading, 332–333
 - ports, 333–334
- Token Ring networking, 339
- troubleshooting, connections, 342
- Ubuntu, 330
- unicast addressing, 338
- VPN, 563, 572–573
 - configuring clients, 573–575
 - configuring servers, 575–577
 - online references, 577
- wireless networks, 361, 382–383
 - configuring, 20
 - interfaces, 340
 - selecting protocols, 363
 - Ubuntu support, 361–363
- New Relic, 299
- new shell programs, running, 239–240
- NewSQL, 511
- newusers command, 224
- NFS, 422
 - configuring
 - clients, 423–424
 - servers, 422–423
 - installing, 422
 - starting/stopping, 422
- nftables, 388
- Nginx web server, 467–469
 - building sources, 469–470
 - configuring, 470–473
 - HTTPS, 476–477

- installing, 469–470
 - modules, 475
 - online references, 477
 - PHP, 474–475
 - uninstalling, 470
 - virtual hosting, 473–474
 - NIC, 338–339
 - 10BASE-T networking, 339
 - 10G Ethernet networking, 340
 - 50G Ethernet networking, 340
 - 100BASE-T networking, 339
 - 1000BASE-T networking, 339
 - fiber-optic networks, 340
 - promiscuous mode, 385
 - security, 385
 - Token Ring networking, 339
 - wireless network interfaces, 340
 - nice command, 177–178
 - Nmap, 330, 382
 - nm-connection-editor, 345–346, 369
 - nonbooting hard disks, saving files from, 324
 - NoSQL databases, 485, 509–510
 - advantages of, 510, 511
 - BaseX, 517
 - Berkeley DB, 512, 579
 - BigTable, 517
 - Cassandra, 513
 - CouchDB, 515–516
 - disadvantages of, 510–511
 - document stores, 514–515
 - etcd, 513
 - FlockDB, 519
 - graph stores, 518
 - HBase, 518
 - HyperGraphDB, 519
 - key/value stores, 512
 - Memcached, 513–514
 - MemcacheDB, 514
 - MongoDB, 516
 - Neo4j, 518
 - NewSQL and, 511
 - online references, 519–520
 - OrientDB, 519
 - Redis, 514
 - Riak, 514
 - Scylla, 514
 - UnQL, 511
 - wide column stores, 517
 - NS records, 591
 - numbering schema, Linux kernels, 24
 - numeric comparisons, 252–253, 256–257
- ## O
- OCaml, 630
 - Office
 - CrossOver Office, 61
 - free/open-source software compatibility, 55
 - LibreOffice compatibility, 55
 - VBA, 55
 - .ogg files, 65, 66
 - .ogv/.ogg files, 75
 - online references
 - Apache web server, 466
 - backups, 324
 - BIND, 595
 - boot process, 278–279
 - cloud computing, 541
 - command line, 139, 165, 200
 - containers, 534
 - desktop environments, 50
 - DNS, 595
 - file sharing, 435
 - games, 89
 - HTTP servers, 482

- Internet connections, 369–370
- kernels, 419–420
- LDAP, 585
- Linux, 23, 28
- multimedia applications, 77
- MySQL, 493, 507
- network printers, 435
- networks, 369–370
- Nginx web server, 477
- NoSQL databases, 519–520
- performance tuning, 396–400
- PostgreSQL, 493, 507
- programming, 618–619
- programming languages, 633–634
- QA, 647
- remote access, 377–378
- security, 391–392
- sets of servers, 545
- shell scripts, 270
- software management, 104
- sound, 65
- Squid, 577
- system monitoring, 299
- system resource management, 104
- testing, 647
- Ubuntu, 22, 28, 30
- Ubuntu development, 643
- user accounts, 224
- virtualization, 529
- VPN, 577
- Wayland, 32
- web browsers, 54
- web server stacks, 441
- X server, 40
- OpenJDK, 604–605
- OpenLDAP. *See* LDAP
- OpenShot Video Editor, 77

- open-source/free software, Office compatibility, 55
- OpenSSH. *See* SSH
- OpenStack, 538–539
- OR operators, 491–492
- optimizing performance, 393–394
 - badblocks command, 397
 - e2fsck command, 397
 - file access time, disabling, 397
 - filesystems, 394–395
 - GNOME, 3933
 - hdparm command, 395–396
 - I/O transfer speeds, 396
 - KDE, 3933
 - kernels, 398–399
 - online references, 396–400
 - storage disks, 394
 - tune2fs command, 396–397
 - Tuned Project, 396–400
- Options directives (Apache web server), 451
- Oracle JDeveloper, 606
- OrientDB, 519
- OSS, 64
- output
 - commands
 - redirecting, 167–170
 - redirecting with shell command line, 236
 - standard input/output, 167, 169
 - paging through, 152–154

P

- PaaS, 537
- packages
 - PPA, 615
 - snap packages, 615–616

- Ubuntu development
 - code packaging, 640–643
 - configuring, 637–638
 - installing, 637–638
- packet writing, Linux command line, DVD creation, 74–75
- paging through output, 152–154
- PAM, 215–216
- parallel, executing jobs in, 181–182
- parameters (positional), 243
 - accessing/retrieving variables, 244
 - example of, 243–244
- parent directories, 118
- partitions
 - planning, Ubuntu installations, 5
 - Ubuntu installations, 5
- passwd command, 114, 209, 224
- passwords, 212, 216, 383–384
 - brute-forcing, 373–374
 - changing in batches, 216–217
 - MySQL root users, 495
 - password files, 212–214
 - shadow passwords, 214–215
 - system password policies, 212
 - Ubuntu, 10
 - user accounts, 202
- patching kernels, 410–411
- pattern-matching, 235
- .pbm files, 69
- pbuilder, 638
- .pcx files, 68
- pdcsch, 226
- PDF, 58
- pdfedit, 58
- pdksh, comparisons of expressions, 250–251
 - file operators, 253–254
 - logical operators, 255
 - numeric comparisons, 252–253
 - strings, 251–252
- performance tuning, 393–394
 - badblocks command, 397
 - e2fsck command, 397
 - file access time, disabling, 397
 - filesystems, 394–395
 - GNOME, 3933
 - hdparm command, 395–396
 - HPC, 598
 - I/O transfer speeds, 396
 - KDE, 3933
 - kernels, 398–399
 - online references, 396–400
 - storage disks, 394
 - tune2fs command, 396–397
 - Tuned Project, 396–400
- periodic basis, full backups on, 306–307
- peripherals
 - digital cameras, 70
 - scanners, 67
- Perl, 630
- permissions
 - directories, 122–127
 - file systems, 120–128
 - altering permissions, 123–124
 - assigning permissions, 121–122
 - changing file groups, 125
 - changing file permissions, 125, 147
 - viewing/modifying default permissions, 124–125
 - files/folders, 204
 - altering permissions, 123–124
 - changing file groups, 125
 - changing permissions, 125, 147
 - viewing/modifying default permissions, 124–125
 - setting with ACL, 127–128
 - sgid permissions, 125–127
 - sticky bit permissions, 126–127
 - suid permissions, 125–127

- pgAdmin, 506
- .pgm files, 69
- Photoshop, 67
- PHP, 474–475, 631
- physical security, 383–384
- PID, 164, 175–176, 237, 274, 566
- ping command, checking Internet connections, 328–329
- ping6 command, checking Internet connections, 328–329
- pipelines (CI/CD), 613
- piping commands, 142–143, 178–180, 237
- PiTiVi, 77
- planning
 - backup strategies
 - assessing needs/resources, 303–304
 - choosing a strategy, 308
 - choosing hardware/media, 308–309
 - evaluating, 304–306
 - full backups on a periodic basis, 306–307
 - full backups with incremental backups, 307
 - home users, 304–305
 - incremental backups, 307
 - inheriting, 306
 - large enterprises, 305
 - mirroring data, 307
 - RAID arrays, 307
 - simple backup strategies, 306
 - small enterprises, 305
 - small offices, 305
 - disaster recovery plans, 390–391
 - partition strategies, Ubuntu installations, 5
 - Ubuntu installations, 1–2
 - flavors of Ubuntu, 2–3
 - partition strategies, 5
 - researching hardware specifications, 2–16
- .png files, 68
- .pnm files, 69
- polite system resets, 194–195
- ports
 - TCP/IP networking, 333–334
 - Ubuntu network ports, 421
 - uplink ports, 342
- positional arguments, 242
- positional parameters, 243
 - accessing/retrieving variables, 244
 - example of, 243–244
- Postfix, 552–556
- PostgreSQL
 - ACID compliance, 493–494
 - command line client, 505–506
 - configuring, 498
 - creating
 - database users, 499–500
 - databases, 499
 - data integrity, 493–494
 - data locking, 492
 - database access, 501–503
 - local GUI clients, 503
 - SSH, 501–502
 - web browsers, 503
 - deleting database users, 500
 - granting/revoking privileges, 500–501
 - initializing data directories, 498
 - MySQL versus, 492–494
 - online references, 493, 507
 - pgAdmin, 506
 - procedural languages, 494
 - retrieving data from tables, 491
 - speed, 492
 - SQL subqueries, 494
 - stored procedures, 494
 - triggers, 494
- POV-Ray, 70

- Power icon (GNOME), 43, 45
- power management, configuring, in Ubuntu, 18–19
- PPA, 615
- .ppm files, 69
- PPP Internet access, configuring, 364
- PPPoE, 366–367
- preformatted DVD, 74
- preinstallation processes, Ubuntu, 1–2
 - flavors of Ubuntu, 2–3
 - hardware specifications, 2–16
- previous commands, rerunning, 196–197
- printing
 - all-in-one (Print/Fax/Scan) devices, 435
 - commands, location of, 164
 - directory sizes, 148
 - disk usage, 148
 - file contents, 144–145
 - file sizes, 148
 - last lines of files, 163–164
 - network printers, 431
 - CUPS GUI, 431–434
 - LAN, 431–432
 - online references, 435
 - server message block printing, 432–433
 - troubleshooting, 434–435
 - resource usage, 175–177
 - server message block printing, 432–433
 - sharing printers, Samba, 428–429
 - Ubuntu printer configurations, 18
- prioritizing
 - processes, 177–178
 - scheduling priorities, 285
- private clouds, 540
- privileges
 - administration privileges, 217
 - granting/revoking with PostgreSQL, 500–501
 - root privileges, 219–222
 - user accounts, 109–111
- /proc directory, 115–117
- procedural languages, relational databases, 494
- processes
 - killing, 283–284
 - listing, 171–173
 - prioritizing, 177–178
 - substituting, 181
- processors, Android mobile development, 608
- productivity applications
 - Celtx, 59
 - CrossOver Office, 61
 - defined, 56
 - gedit, 59
 - Heimer, 61
 - Kile, 61
 - LaTeX, 60
 - LibreOffice
 - brief history of, 57–58
 - components of, 56–57
 - Office compatibility, 55
 - VBA, 55
 - LyX, 60
 - online references, 61
 - pdfedit, 58
 - Publican, 59–60
 - Texmaker, 60
 - users (typical), 56
 - Windows, 61
 - Wine, 61
 - XML Copy Editor, 60
- productivity suites, LibreOffice, Office compatibility, 55
- programming
 - Android mobile development, 607
 - Android Runtime, 608
 - Android Studio, 609

- Application Framework, 608–609
- applications, 609
- ARM processors, 608
- Google Play, 607
- Linux kernels, 608
- RISC processors, 608
- Bikeshed, 616–618
- C, 597–599
 - autoconf command, 601–602
 - building programs, 599–601
 - checking source code, 602–603
 - configuring code, 601–602
 - debugging tools, 602–603
 - dependency checking, 599–600
 - GCC, 603–604
 - gdb command, 603
 - gprof command, 603
 - macros, 600–601
 - makefile targets, 600–601
 - makefiles, 599–600
 - splint command, 602–603
 - symbolic debugging, 603
 - tracking function time, 603
- C++597–598, 599
 - autoconf command, 601–602
 - building programs, 599–601
 - checking source code, 602–603
 - configuring code, 601–602
 - debugging tools, 602–603
 - dependency checking, 599–600
 - GCC, 603–604
 - gdb command, 603
 - gprof command, 603
 - macros, 600–601
 - makefile targets, 600–601
 - makefiles, 599–600
 - splint command, 602–603
 - symbolic debugging, 603
 - tracking function time, 603
- Chaos Engineering, 614
- CI/CD tools, 613–614
- GCC, 597
- graphical development tools, 605
 - Eclipse, 606
 - Glade, 607
 - IDE, 605–606
 - KDevelop Client, 606–607
 - NetBeans, 606
 - Oracle JDeveloper, 606
 - SDK, 606
 - Visual Studio Code, 606
- Java programming, 604
 - JVM, 604
 - OpenJDK, 604–605
- languages, 621–622
 - Ada, 622
 - Clojure, 622–623
 - COBOL, 623–624
 - D, 624
 - Dart, 624
 - Elixir, 625
 - Elm, 625
 - Erlang, 625
 - Forth, 626
 - Fortran, 626
 - Go, 626–627
 - Groovy, 627
 - Haskell, 627
 - Java, 627–628
 - JavaScript, 628
 - Kotlin, 628–629
 - Lisp, 629
 - Lua, 629
 - Mono, 629–630

- OCaml, 630
- online references, 633–634
- Perl, 630
- PHP, 631
- Python, 631
- Raku, 631
- Ruby, 631–632
- Rust, 632
- Scala, 632
- Scratch, 632–633
- Vala, 633
- online references, 618–619
- snap packages, 615–616
- software project management
 - Bazaar, 611–612
 - Git, 610–611
 - Subversion, 612–613
- Ubuntu Make, 615
- version control systems, 609–610
- promiscuous mode (NIC), 385
- proxy servers
 - defined, 563–564
 - Squid, 563–564
 - ACL, 565–569
 - configuring, 570–572
 - configuring clients, 564–565
 - installing, 564
 - online references, 577
 - specifying client IP addresses, 569–570
- ps command, 171–173
- psql command, 506
- public clouds, 538, 539
- Publican, 59–60
- PulseAudio, 64
- Puppet, 543–544
- /pwd command, 120
- PXE, boot process, 274
- Python, 631

Q

- QA
 - Bug Squad, 647
 - online references, 647
 - Ubuntu QA Teams, 646–647
- Qmail, 549–550
- .qt files, 75
- quotas (disk), 287
 - configuring, 223
 - implementing, 222–223
 - managing, 222

R

- Radiance, 70
- RAID arrays, 307
- Raku, 631
- RAM disk images (kernels), creating, 418
- RARP, 338
- .raw files, 65
- Raymond, Eric, 141–142
- rcp command, 372
- RDP, VirtualBox, 528
- rebooting systems, 138–139
- recompiling kernels, 408–409
- recording/editing sound, 76
 - Ardour, 76
 - Audacity, 76
 - Cecilia, 76
 - LMMS, 76
 - Mixxx, 76
 - Rosegarden, 76
- recovery
 - data, 323
 - GRUB2 boot loaders, restoring, 323–324

- saving files from nonbooting hard disks, 324
 - Ubuntu Rescue disc, 323
 - disaster recovery plans, 390–391
- recovery mode (GRUB), 196
- reddit.com, 272
- redirecting input/output of commands, 167–170, 236
- Redis, 514
- references (online)
 - Apache web server, 466
 - BIND, 595
 - boot process, 278–279
 - cloud computing, 541
 - command line, 139, 165, 200
 - containers, 534
 - desktop environments, 50
 - DNS, 595
 - file sharing, 435
 - games, 89
 - HTTP servers, 482
 - Internet connections, 369–370
 - kernels, 419–420
 - LDAP, 585
 - Linux, 23, 28
 - multimedia applications, 77
 - MySQL, 493, 507
 - network printers, 435
 - networks, 369–370
 - Nginx web server, 477
 - NoSQL databases, 519–520
 - online references, 324
 - performance tuning, 396–400
 - PostgreSQL, 493, 507
 - programming, 618–619
 - programming languages, 633–634
 - QA, 647
 - remote access, 377–378
 - security, 391–392
 - sets of servers, 545
 - shell scripts, 270
 - software management, 104
 - sound, 65
 - Squid, 577
 - system monitoring, 299
 - system resource management, 104
 - testing, 647
 - Ubuntu, 22, 28, 30
 - Ubuntu development, 643
 - user accounts, 224
 - virtualization, 529
 - VPN, 577
 - Wayland, 32
 - web browsers, 54, 61
 - web server stacks, 441
 - X server, 40
- regular expressions, 235
 - comparisons of expressions, 250–251
 - file operators, 253–254, 257–258
 - logical operators, 255, 259
 - numeric comparisons, 252–253, 256–257
 - strings, 251–252, 255–256
 - directories, 133
- reinstalling
 - GRUB, 195–196
 - Ubuntu, 196
- REISUB, polite system resets, 194–195
- relational databases, 485
 - DBA, 483–484
 - graphical database clients, 506
 - MySQL, 483
 - ACID compliance, 493–494
 - command line client, 504–505
 - configuring, 494–495

- creating databases, 496–497
 - data integrity, 493–494
 - data locking, 492
 - database access, 501–503
 - inserting data into tables, 490
 - mysql command, 506
 - mysqladmin command, 506
 - mysqldump command, 506
 - MySQLGUI, 506
 - online references, 493, 507
 - PostgreSQL versus, 492–494
 - procedural languages, 494
 - retrieving data from tables, 490–492
 - root user passwords, 495
 - speed, 492
 - SQL subqueries, 494
 - stored procedures, 494
 - triggers, 494
- operation of, 486–487
- PostgreSQL
- ACID compliance, 493–494
 - command line client, 505–506
 - configuring, 498
 - creating database users, 499–500
 - creating databases, 499
 - data integrity, 493–494
 - data locking, 492
 - database access, 501–503
 - deleting database users, 500
 - granting/revoking privileges, 500–501
 - initializing data directories, 498
 - MySQL versus, 492–494
 - online references, 493, 507
 - pgAdmin, 506
 - procedural languages, 494
 - retrieving data from tables, 491
 - speed, 492
 - SQL subqueries, 494
 - stored procedures, 494
 - triggers, 494
- SQL, 487–492
- creating tables, 488–489
 - inserting data into tables, 489–490
- remote access
- copying files between machines, 372–373
 - Guacamole, 377
 - key-based logins, 373–375
 - online references, 377–378
 - SSH server, 371–375
 - VNC, 375–377
- remote computers, command line, logins, 108–109
- rename command, 161
- renaming files/folders, 161
- repeat statements, 263
- repeatedly running jobs, 229–231
- repeating text, 148–149
- repositories
- software repositories, configuring, 15–17
 - Ubuntu repositories, compiling software, 101–102
- Require directive (Apache web server), 452
- rerunning previous commands, 196–197
- resetting systems, polite system resets, 194–195
- resources
- assessing, backup strategies, 303–304
 - usage, printing, 175–177
- restoring
- files from archives, 311–312
 - GRUB2 boot loaders, 323–324
- retrieving data from SQL tables, 490–492
- revoking/granting privileges, PostgreSQL, 500–501

- Rhythmbox music application, 66
- Riak, 514
- ripping audio, 71
- RISC processors, 608
- rm command, 130–131, 161–162
- rmdir command, 129–130
- rmmod command, 406
- root accounts, 110–111, 133–134
- root privileges, 219–222
- root prompt, dangers of working in, 15
- root user accounts, 202–203
- root zones, 589
- Rosegarden, 76
- rotating log files, 289–291
- route command, 369
- routers, 343
- rsync command, 319–320
- rtcwake command, 231–233
- Ruby, 631–632
- runlevels, 271–272, 275, 278
- Runtime (Android), 608
- runtime servers, Apache web server
 - apache2.conf files, 447–449
 - configuring, 446–449
 - DirectoryIndex directive, 449
 - DocumentRoot directive, 449
 - GID, 448
 - Group directive, 448
 - Listen directive, 447
 - ServerAdmin directive, 448
 - ServerName directive, 448–449
 - ServerRoot directive, 447
 - UID, 448
 - User directive, 448
 - UserDir directive, 449
- Rust, 632

S

- SaaS, 537
- SaltStack, 544–545
- Samba, 424
 - configuring, 425, 427
 - /etc/samba/smb.conf files, 426–429
 - installing, 425–426
 - mounting, 430–431
 - sharing
 - home directories, 427–428
 - printers, 428–429
 - smbclient command, 430
 - smbd daemon, starting/stopping, 429–430
 - smbstatus command, 430
 - testing, 429
- saving, files from nonbooting hard disks, 324
- /sbin directory, 114
- Scala, 632
- scanners, 67, 435
- scheduling
 - backups, full backups on a periodic basis, 306–307
 - priorities, 285
 - tasks, 226
 - automating tasks with scripts, 244–246
 - for later, 227–229
 - running jobs repeatedly, 229–231
 - waking computers from sleep, 231–233
- Scorched 3D, 83–84
- scp command, 372–373
- Scratch, 632–633
- screen images, capturing, 69
- Screen section (xorg.conf files), 38–39
- script kiddies, 380
- scripting
 - autocracking scripts, 380
 - automating tasks, 244–246

- break statements, 269
 - case statements, 267–268
 - Celx, 59
 - comparisons of expressions, 250–251
 - file operators, 253–254, 257–258
 - logical operators, 255, 259
 - numeric comparisons, 252–253, 256–257
 - strings, 251–252, 255–256
 - confining to directories, 198–199
 - endless loops, 261–262
 - executing scripts, 237–239
 - exit statements, 269
 - expr statements, 266
 - for statements, 259–261, 263
 - functions, 269–270
 - if statements, 265–266
 - interpreting scripts through specific shells, 240–241
 - new shell programs, 239–240
 - online references, 270
 - positional arguments, 242
 - positional parameters, 243
 - accessing/retrieving variables, 244
 - example of, 243–244
 - repeat statements, 263
 - select statements, 264
 - shift statements, 264
 - special shell characters
 - ^ special shell character, 250
 - ", 248–249
 - ' special shell character, 249
 - backslash, 249–250
 - list of, 247–248
 - storing scripts for system-wide access, 240
 - strings with embedded spaces, resolving, 248–249
 - variables, 242
 - accessing values, 243
 - accessing/retrieving with positional parameters, 244
 - assigning values, 242
 - built-in variables, 246–247
 - unexpanded variables, 249
 - while statements, 261, 262
 - writing scripts, 237–239
- Scylla, 514
- SDK, 606
- searches
 - DNS search orders, 355
 - input strings, 151–152
- security
 - access control, Apache web server, 452–455
 - Apache web server, 444, 452
 - AppArmor, 388–390
 - applications, 388–390
 - attacks, 380–381
 - autocracking scripts, 380
 - crackers, 380
 - external attacks, 379–380
 - hackers, 380
 - internal attacks, 379–380
 - script kiddies, 380
 - viruses, 385
 - war driving, 383
 - worms, 380
 - authentication
 - Apache web server, 452–455
 - PAM, 215–216
 - brute-forcing, 373–374
 - checklist, 381
 - configuring, 384–385
 - devices, 385
 - disaster recovery plans, 390–391
 - encrypted files, 318

- firewalls, 386–388
 - iptables, 388
 - nftables, 388
 - UFW, 386–387
- genprof, 389
- kernels, 379
- nftables, 388
- NIC, 385
- online references, 391–392
- passwords, 212, 216, 383–384
 - brute-forcing, 373–374
 - changing in batches, 216–217
 - MySQL root users, 495
 - password files, 212–214
 - shadow passwords, 214–215
 - system password policies, 212
 - Ubuntu installations, 10
 - user accounts, 202
- physical security, 383–384
- proxy servers. *See also* Squid, 563–564
- spoofing attacks, 593
- Tripwire, 384–385
- UFW, 386–387
- viruses, 385
- VPN, 563, 572–573
 - configuring clients, 573–575
 - configuring servers, 575–577
 - online references, 577
- vulnerability assessments, 381–382
- wireless networks, 382–383
- sed text editor, 189–191
- select statements, 264
- SELinux, 388
- semistructured data, 515
- sequence, running commands in, 180–181
- ServerAdmin directive (Apache web server), 448
- ServerLayout section (xorg.conf files), 35
- ServerName directive (Apache web server), 448–449
- ServerRoot directive (Apache web server), 447
- servers
 - Apache web server
 - access control, 452, 455
 - configuring, 446–449, 450–452
 - development of, 443–444
 - file system authentication, 452–455
 - .htaccess configuration files, 450–452, 469
 - HTTPS, 464–466
 - installing, 444
 - logging, 463–464
 - modules, 455–461
 - MPM, 449–450
 - online references, 466
 - overview of, 444
 - runtime servers, 446–449
 - security, 444, 452–455
 - starting/stopping, 444–446
 - threads, 468
 - virtual hosting, 461–462
 - DHCP servers, 358–359
 - DNS servers, 588, 589–590, 593–595
 - enterprise servers, monitoring, 298–299
 - FQDN, 580
 - HTTP servers
 - Caddy, 482
 - Cherokee, 480–481
 - Jetty, 481
 - lighttpd, 479–480
 - online references, 482
 - thttpd, 481–482
 - Tomcat, 482
 - Wildfly, 482
 - Yaws, 480

- LDAP servers, configuring, 580
- message block printing, 432–433
- name servers, 352–355
- NFS servers, 422–423
- Nginx web server, 467–469
 - building sources, 469–470
 - configuring, 470–473
 - HTTPS, 476–477
 - installing, 469–470
 - modules, 475
 - online references, 477
 - PHP, 474–475
 - uninstalling, 470
 - virtual hosting, 473–474
- proxy servers, defined. *See also* Squid, 563–564
- runtime servers, Apache web server, 446–449
- sets of servers, managing, 543
 - Ansible, 544
 - CFEngine, 545
 - Chef, 544
 - Juju, 545
 - Landscape, 545
 - online references, 545
 - Puppet, 543–544
 - SaltStack, 544–545
- Squid, 563–564
 - ACL, 565–569
 - configuring, 570–572
 - configuring clients, 564–565
 - installing, 564
 - online references, 577
 - specifying client IP addresses, 569–570
- SSH server
 - configuring, 371–372
 - copying files between machines, 372–373
 - ftp command, 372
 - key-based logins, 373–375
 - scp command, 372–373
 - sftp command, 373
 - ssh-keygen command, 373–375
- VPN servers, configuring, 575–577
- web servers
 - C10K problem, 467–468
 - event handlers, 468–469
 - threads, 468
- X server, 32, 33
 - /usr directory, 33
 - benefits of, 33–34
 - components of, 33–34
 - display manager, 39
 - display managers, 33
 - distributed processing, 32–33
 - online references, 40
 - starting, 39
 - terminal clients, 33–34
 - window manager, 33–34
 - xorg.conf files, 33–39
- server stacks, 437
 - LAMP, 437–439
 - LEMP, 439
 - MEAN, 437
 - online references, 441
- services
 - boot process
 - controlling, 278
 - running services, 271–272
 - starting/stopping services, 275–278
 - Startup Applications Preferences, 278
 - /etc/services files, 351
 - microservice architectures, 531
 - naming services, 351–352
 - units, listing, 276

- session writing, DVD creation, 74
- set group ID permissions (suid), 125–127
- set user ID permissions (suid), 125–127
- sets of servers, managing, 543
 - Ansible, 544
 - CFEngine, 545
 - Chef, 544
 - Juju, 545
 - Landscape, 545
 - online references, 545
 - Puppet, 543–544
 - SaltStack, 544–545
- sftp command, 373, 383
- sgid permissions, 125–127
- shadow passwords, 214–215
- shared data, /usr directory, 117
- sharing
 - files/folders
 - NFS, 422–424
 - online references, 435
 - Samba, 424–431
 - home directories with Samba, 427–428
 - printers, Samba, 428–429
 - Ubuntu desktops, 424
- shells. *See also* command line, 233
 - ash, 226
 - bash, 226, 250–255
 - changing, 226
 - command line, 233–234
 - background processes, 237
 - pattern-matching, 235
 - piping commands, 237
 - dash, 226
 - defined, 225–226
 - expressions, 235
 - GNOME Shell, 43–45
 - ksh, 226
 - Linux, 25
 - local bash shells, configuring, 640
 - new shell programs, 239–240
 - pdcksh, 226
 - pdcksh, comparisons of expressions, 250–255
 - scripting
 - ˘ special shell character, 250
 - " , 248–249
 - ' special shell character, 249
 - automating tasks, 244–246
 - backslash, 249–250
 - break statements, 269
 - case statements, 267–268
 - comparisons of expressions, 250–259
 - endless loops, 261–262
 - executing scripts, 237–239
 - exit statements, 269
 - expr statements, 266
 - functions, 269–270
 - if statements, 265–266
 - interpreting through specific shells, 240–241
 - new shell programs, 239–240
 - online references, 270
 - positional arguments, 242
 - positional parameters, 243–244
 - repeat statements, 263
 - resolving strings with embedded spaces, 248–249
 - select statements, 264
 - shift statements, 264
 - special shell characters, 247–248
 - for statements, 259–261
 - storing scripts for system-wide access, 240
 - until statements, 263
 - variables, 242–243, 244, 246–247, 249

- while statements, 261, 262
 - writing scripts, 237–239
- tcsh, 226, 255–259
- types of, 225–226
- zsh, 226
- shift statements, 264
- shortcuts, commands, 198
- Shotcut, 77
- Shotwell Photo Manager, 66, 71
- Show Applications icon (GNOME), 43, 44
- shred command, 155
- shutdown command, 137–139
- shutting down
 - systems, 137–138
 - Ubuntu, 11–12
- SignalFX, 299
- similarities in files, finding, 170–171
- simple backup strategies, 306
- Simple Mail Transfer Protocol (SMTP), 547
- Simple Scan, 67
- sleep, waking computers from, 231–233
- small enterprises, backup strategies, 305
- small offices, backup strategies, 305
- smart gateways, 343
- smbclient command, 430
- smbd daemon, starting/stopping, 429
- smbstatus command, 430
- snap packages, 615–616
- SnapCraft, 616
- Snappy package manager, 103–104
- Snappy Playpen, 616
- snaps, Snappy package manager, 103–104
- SOA records, 592–593
- soft links. *See* symlinks
- software
 - backup software, 309–316
 - bundles, Snappy package manager, 103–104
 - compiling
 - from source, 100, 101–102
 - from tarballs, 100–101
 - from Ubuntu repositories, 101–102
 - DHCP software, installing/configuring, 358–359
 - finding, with APT, 98–99
 - free/open-source software, MS Office compatibility, 55
 - libraries, Android mobile development, 608
 - Linux distributions, included software, 24–25
 - management tools
 - APT, 95–100
 - apt-get, 99–100
 - compiling software, 100–102
 - configuration management, 102–104
 - online references, 104
 - Software Updater, 94–95
 - Synaptic, 92–94
 - Ubuntu Software, 91
 - open-source/free software, MS Office compatibility, 55
 - project management
 - Bazaar, 611–612
 - Git, 610–611
 - Subversion, 612–613
 - repositories, configuring, 15–17
 - Snappy package manager, 103–104
 - updating, 12–14
- Software Updater, 12–14, 94–95
- sort command, 162–163
- sorting, file contents, 162–163
- sound, 65
 - ALSA, 64
 - CD/DVD burning applications
 - Brasero, 71
 - Linux command line, CD creation, 72–73

- Linux command line, DVD creation, 73–75
- Ubuntu support, 71
- converting files, 65
- .flac files, 65
- .mp3 files, 65
- .mp4 files, 65
- music applications, 65
 - Banshee, 66
 - Rhythmbox, 66
 - Sound Juicer, 66
- .ogg files, 65, 66
- online references, 65
- OSS, 64
- PulseAudio, 64
- recording/editing, 76
 - Ardour, 76
 - Audacity, 76
 - Cecilia, 76
 - LMMS, 76
 - Mixxx, 76
 - Rosegarden, 76
- recording/editing applications, sound, 76
- ripping audio, 71
- sound cards, 63–64
- sox command, 65
- UNIX and, 63–64
- .wav files, 65
- Sound Juicer, 66
- source, compiling software from, 100, 101–102
- source tree (kernels), 403–405
- sources (kernels), obtaining, 409–410
- sox command, 65
- spaces
 - embedded spaces, resolving strings with, 248–249
 - as special shell characters, 248
- Spamassassin, 561
- special characters
 - & operator, 173–174, 237, 247, 527
 - && operators, combining commands, 180, 198
 - * special shell character, 247
 - \ special shell character, 249–250
 - ` special shell character, 247, 250
 - [] special shell character, 248
 - [a, z] special shell character, 248
 - [a-z] special shell character, 248
 - [az] special shell character, 248
 - \$, 235
 - \$? built-in variable, 247
 - \$ special shell character, 247
 - \$# built-in variable, 246
 - \$* built-in variable, 247
 - \$0 built-in variable, 247
 - !! operator, 196–197
 - != operator, 491
 - < special shell character, 247
 - << operator, 236
 - << special shell character, 248
 - > special shell character, 247
 - >> special shell character, 247
 - | (pipe) special shell character, 247
 - || operator, 198, 491
 - # special shell character, 247
 - ? special shell character, 247
 - " special shell character, 248–249
 - ' special shell character, 249
- speed, relational databases, 492
- Speed Dreams, 87
- splint command, 602–603
- Splunk, 299
- spoofing attacks, 593
- SQL, 487–488
 - != operator, 491
 - MySQL, 483

- ACID compliance, 493–494
- command line client, 504–505
- configuring, 494–495
- creating databases, 496–497
- data integrity, 493–494
- data locking, 492
- database access, 501–503
- inserting data into tables, 490
- mysql command, 506
- mysqladmin command, 506
- mysqldump command, 506
- MySQLGUI, 506
- online references, 493, 507
- PostgreSQL versus, 492–494
- procedural languages, 494
- retrieving data from tables, 490–492
- root user passwords, 495
- speed, 492
- SQL subqueries, 494
- stored procedures, 494
- triggers, 494
- NoSQL databases, 485
- PostgreSQL
 - ACID compliance, 493–494
 - command line client, 505–506
 - configuring, 498
 - creating database users, 499–500
 - creating databases, 499
 - data integrity, 493–494
 - data locking, 492
 - database access, 501–503
 - deleting database users, 500
 - granting/revoking privileges, 500–501
 - initializing data directories, 498
 - MySQL versus, 492–494
 - online references, 493, 507
 - pgAdmin, 506
 - procedural languages, 494
 - retrieving data from tables, 491
 - speed, 492
 - SQL subqueries, 494
 - stored procedures, 494
 - triggers, 494
 - subqueries, 494
 - tables
 - creating, 488–489
 - inserting data, 489–490
- Squid, 563–564
 - ACL, 565–569
 - client IP addresses, 569–570
 - configuring, 564–565, 570–572
 - installing, 564
 - online references, 577
- Squirrelmail, 561
- SSH, database access, 501–502
- ssh command, 369, 383
- SSH keys, 638, 639
- SSH server
 - configuring, SSH server, 371–372
 - copying files between machines, 372–373
 - ftp command, 372
 - key-based logins, 373–375
 - scp command, 372–373
 - sftp command, 373
 - ssh-keygen command, 373–375
- ssh-keygen command, 373–375
- stacks, 437
 - LAMP, 437–439
 - LEMP, 439
 - MEAN, 437
 - online references, 441
- standard input/output of commands, 169
- starting/stopping
 - Apache web server, 444–446
 - NFS, 422

- services, 275–278
- smbd daemon, 429–430
- VM, 526
- Startup Applications Preferences, 278
- statements
 - break statements, 269
 - case statements, 267–268
 - exit statements, 269
 - expr statements, 266
 - for statements, 259–261
 - if statements, 265–266
 - repeat statements, 263
 - select statements, 264
 - shift statements, 264
 - for statements, 259–261
 - until statements, 263
 - while statements, 261, 262
- statistics, user usage, 212
- stderr command, 169–170
- stdin command, 169–170
- stdout command, 169–170
- Steam, 81
- stereotypes, users, 205
- sticky bit permissions, 126–127
- stopping/starting
 - Apache web server, 444–446
 - NFS, 422
 - services, 275–278
 - smbd daemon, 429–430
 - VM, 526
- storage
 - backups
 - cloud storage, 309
 - external hard drives, 308
 - NAS, 308
 - network storage, 308
 - tape drive backups, 309
 - disks, performance tuning, 394
 - drives, Ubuntu installations, 9
 - temporary file storage, /tmp directories, 117
- stored procedures
 - MySQL, 494
 - relational databases, 494
- strings
 - command substitution, 250
 - comparisons, 251–252, 255–256
 - with embedded spaces, resolving, 248–249
 - input strings, searches, 151–152
- su command, 217–219, 224
- subnet masks, 337–338
- subnetting, 337–338
- subqueries (SQL), relational databases, 494
- substitute user accounts, 217–218
- substituting
 - commands, 250
 - processes, 181
- Subversion, software project management, 612–613
- sudo command, 14–15, 110–111, 134–136, 219–222, 224
- sudoers command, 114
- suid permissions, 125–127
- super user accounts, 109–111, 133–134, 202–203, 217–218
- SuperTux, 84–85
- support, Ubuntu online references, 22
- .svg files, 68
- switches, 342
- symbolic debugging, 603
- symbols. *See also* wildcard characters
 - & operator, 237
 - background processes, running, 173–174
 - special shell character, 247
 - VirtualBox commands, 527

- && operators, combining commands, 180, 198
- * special shell character, 247
- \ special shell character, 249–250
- ` special shell character, 247, 250
- [] special shell character, 248
- [a, z] special shell character, 248
- [a-z] special shell character, 248
- [az] special shell character, 248
- \$, 235
- \$? built-in variable, 247
- \$ special shell character, 247
- \$# built-in variable, 246
- \$* built-in variable, 247
- \$0 built-in variable, 247
- !! operator, 196–197
- != operator, 491
- < special shell character, 247
- << operator, 236
- << special shell character, 248
- > special shell character, 247
- >> special shell character, 247
- | (pipe) special shell character, 247
- || operator, 198, 491
- # special shell character, 247
- ? special shell character, 247
- " special shell character, 248–249
- ' special shell character, 249
- symlinks, 154–156
- Synaptic, 92–94
- synchronizing, filesystems, 396
- Sysadmin, DevOps versus, 535–536
- sysctl command, 398–399
- sysrq.txt files, 404
- System Monitor, 292
- system resource management
 - APT, 95
 - apt-get versus, 99–100
 - day-to-day usage, 95–98
 - finding software, 98–99
 - apt-get, APT versus, 99–100
 - compiling software
 - from source, 100, 101–102
 - from tarballs, 100–101
 - from Ubuntu repositories, 101–102
 - configuration management, 102
 - dotdee, 102–103
 - Snappy package manager, 103–104
 - Ubuntu Core, 103
 - online references, 104
 - Software Updater, 94–95
 - Synaptic, 92–94
 - Ubuntu Software, 91
- system settings, Ubuntu settings, configuring, 17–18
- system user accounts, 203–204
- systemd command, 275, 277–278, 532
 - service-related commands, 276–277
 - state-related commands, 276
 - targets, 275–276
- systems
 - administration privileges, 217
 - listing information, 158–159
 - management tools, 292
 - Conky, 292–297
 - gnome-nettool, 297
 - System Monitor, 292
 - vncviewer, 297
 - Wireshark, 298
- monitoring
 - checking log files, 287–289
 - Conky, 292–297
 - console-based monitoring, 281–283
 - disk quotas, 287
 - displaying free hard disk space, 286–287
 - displaying used memory, 286

- enterprise servers, 298–299
 - graphical process tools, 292–298
 - KDE monitoring tools, 298
 - killing processes, 283–284
 - online references, 299
 - priority scheduling, 285
 - rotating log files, 289–291
 - system management tools, 292–298
 - System Monitor, 292
 - user activity, 211–212
 - password policies, 212
 - polite resets, 194–195
 - rebooting, 138–139
 - rescue, 323
 - restoring GRUB2 boot loaders, 323–324
 - saving files from nonbooting hard disks, 324
 - Ubuntu Rescue disc, 323
 - shutting down, 137–138
 - storing shell scripts for system-wide access, 240
 - SystemV runlevels, 275–276
- ## T
- tables (SQL)
 - creating, 488–489
 - inserting data, 489–490
 - tail command, 163–164
 - tape drive backups, 309
 - tar command, 192, 310
 - compressed files, 318
 - copying files, 317–318
 - encrypted files, 318
 - full backups, 310–311
 - incremental backups, 310–311
 - restoring files from archives, 311–312
 - tarballs, compiling software from, 100–101
 - tasks, scheduling, 226
 - automating tasks with scripts, 244–246
 - for later, 227–229
 - running jobs repeatedly, 229–231
 - waking computers from sleep, 231–233
 - tcpdump command, 385
 - TCP/IP networking, 330
 - addressing, 331–332
 - IP masquerading, 332–333
 - ports, 333–334
 - tcsh, comparisons of expressions, 226, 255
 - file operators, 257–258
 - logical operators, 259
 - numeric comparisons, 256–257
 - strings, 255–256
 - temporary file storage, /tmp directories, 117
 - terminals. *See also* command line
 - clients, X server, 33–34
 - multiple terminal operation, byobu, 192–193
 - testing
 - online references, 647
 - Samba, 429
 - Ubuntu
 - community teams, 645–646
 - Ubuntu Testing Teams, 646
 - testparm command, 429
 - TeX documents
 - Kile, 61
 - LaTeX, 60
 - LyX, 60
 - Texmaker, 60
 - Texmaker, 60
 - text
 - editors, 185, 186
 - awk, 189–191
 - emacs, 185, 188–189

- gedit, 186
- kate, 186
- kedit, 186
- nano, 185, 186–187
- sed, 189–191
- vi, 185, 187–188
- vim, 185
- repeating, 148–149
- text-based logins, command line, 107–108
- threads, 468
- thttpd, 481–482
- thumb drives, Ubuntu installations, 3, 6–11
- Thunderbird, LDAP client configuration, 584
- .tif files, 68
- time warps, 212
- time/date settings
 - date command, 19
 - hwclock command, 19–20
 - Ubuntu, 19–20
- TLD, 588–589
- /tmp directories, 117
- Token Ring networking, 339
- Tomcat, 482
- top command, 175–177
- Totem Movie Player, 76
- touch command, 128–129
- traceroute command, checking Internet connections, 329
- traceroute6 command, checking Internet connections, 329
- triggers, relational databases, 494
- Tripwire, 384–385
- troubleshooting
 - boot process, 278–279
 - bugs
 - Bug Squad, 647
 - Ubuntu development, 640–643
 - Internet connections, 368–369

- kernels, 418–419
- network printers, 434–435
- networks, connections, 342
- runlevels, 278
- sudo command, 134–136
- Ubuntu
 - bootups, 195–196
 - post-configuration problems, 21–22
- tune2fs command, 396–397
- Tuned Project, 396–400
- TXT records, 593
- type name command, 160

U

- Ubuntu, 27
 - 32-bit Ubuntu, 4–5
 - 64-bit Ubuntu, 4–5
 - Android mobile development, 607
 - Android Runtime, 608
 - Android Studio, 609
 - Application Framework, 608–609
 - applications, 609
 - ARM processors, 608
 - Google Play, 607
 - Linux kernels, 608
 - RISC processors, 608
- bash, 226
- Bikeshed, 616–618
- bootups, troubleshooting, 195–196
- Budgie, 49
- business applications, 27–28
- C programming
 - autoconf command, 601–602
 - building programs, 599–601
 - checking source code, 602–603

- configuring code, 601–602
- debugging tools, 602–603
- dependency checking, 599–600
- GCC, 603–604
- gdb command, 603
- gprof command, 603
- macros, 600–601
- makefile targets, 600–601
- makefiles, 599–600
- splint command, 602–603
- symbolic debugging, 603
- tracking function time, 603
- C++ programming
 - autoconf command, 601–602
 - building programs, 599–601
 - checking source code, 602–603
 - configuring code, 601–602
 - debugging tools, 602–603
 - dependency checking, 599–600
 - GCC, 603–604
 - gdb command, 603
 - gprof command, 603
 - macros, 600–601
 - makefile targets, 600–601
 - makefiles, 599–600
 - splint command, 602–603
 - symbolic debugging, 603
 - tracking function time, 603
- CD/DVD burning applications, 71
- Chaos Engineering, 614
- CI/CD tools, 613–614
- cloud computing, 535
 - AWS, 539
 - Azure, 539
 - considerations for using, 538
 - Google Cloud, 539
 - hybrid clouds, 539–540
 - laaS, 537
 - install instructions, 536
 - Juju, 540–541, 545
 - Landscape, 541
 - MaaS, 537
 - Mojo, 541
 - OpenStack, 538–539
 - PaaS, 537
 - private clouds, 540
 - public clouds, 538, 539
 - reasons for using, 536–537
 - SaaS, 537
 - Ubuntu Cloud, 536
 - VM, 536
- configuring
 - date/time settings, 19–20
 - power management, 18–19
 - system settings, 17–18
 - troubleshooting post-configuration problems, 21–22
 - wireless networks, 20
- database commands, 506
- date/time settings, 19–20
- desktop environments, changing, 39
- desktops, sharing, 424
- development
 - Bazaar, 639
 - code packaging, 640–643
 - fixing bugs, 640–643
 - GPG keys, 637–639
 - introduction to, 636–637
 - Launchpad, 614–615, 638–640
 - local bash shells, 640
 - online references, 643
 - packages, 637–638, 640–643
 - pbuilder, 638
 - SSH keys, 638, 639
- digital cameras, 70
- distributions, versions of, 27

- documentation, 28, 30
- dual-boot systems, 6
- DVD/CD burning applications, 71
- finding files/folders, 12
- graphical development tools, 605
 - Eclipse, 606
 - Glade, 607
 - IDE, 605–606
 - KDevelop Client, 606–607
 - NetBeans, 606
 - Oracle JDeveloper, 606
 - SDK, 606
 - Visual Studio Code, 606
- home applications, 28
- installing, 196
 - 32-bit Ubuntu, 4–5
 - 64-bit Ubuntu, 4–5
 - backups, 7
 - boot loaders, 5–6
 - downloading updates during installation, 8
 - dual-boot systems, 6
 - DVD installations, 2, 6–11
 - erasing hard disks during installation, 9
 - flavors of Ubuntu, 2–3
 - language selection, 7–8
 - Mac hardware compatibility, 3
 - partitions, 5, 9
 - password creation, 10
 - preinstallation process, 1–2
 - researching hardware specifications, 2–16
 - storage drives, 9
 - UEFI, 6
 - USB thumb drive installations, 3, 6–11
- Java programming, 604
 - JVM, 604
 - OpenJDK, 604–605
- kernels, 401–402
- Kylin, 50
- MATE, 48–49
- networks, 330
 - connectivity commands, 369
 - ports, 421
 - printers, 431–436
- Nginx web server installations, 469
- online references, 22
- partitions, installing, 5, 9
- passwords, 10
- power management, configuring, 18–19
- preinstallation process, 1–2
 - flavors of Ubuntu, 2–3
 - researching hardware specifications, 2–16
- printers, configuring, 18
- proxy servers. *See* Squid
- QA
 - Bug Squad, 647
 - Ubuntu QA Teams, 646–647
- reinstalling, 196
- repositories, compiling software, 101–102
- root prompt, dangers of working in, 15
- scanners, 67
- shutting down, 11–12
- snap packages, 615–616
- SnapCraft, 616
- Snappy Playpen, 616
- software project management
 - Bazaar, 611–612
 - Git, 610–611
 - Subversion, 612–613
- software repositories, configuring, 15–17
- Software Updater, 12–14
- storage drives, Ubuntu installations, 9
- sudo command, 14–15
- system settings, configuring, 17–18

- testing
 - community teams, 645–646
 - Ubuntu Testing Teams, 646
- troubleshooting, post-configuration problems, 21–22
- Ubuntu Make, 615
- Ubuntu Rescue disc, 323
- updating
 - first updates, 11
 - during installation, 8
 - software, 12–14
- user account management, 223–224
- version control systems, 609–610
- virtualization, 521–523
 - bridged networks, 524–525
 - KVM, 523–526
 - VirtualBox, 527–528
 - VM, 522, 525–526, 536
- web resources, 22
- window managers, changing, 39
- wireless networks, 20, 361–363
- X server, 32–33
 - benefits of, 33–34
 - components of, 33–34
 - display manager, 39
 - display managers, 33
 - distributed processing, 32–33
 - online references, 40
 - starting, 39
 - terminal clients, 33–34
 - /usr directory, 33
 - window manager, 33–34
 - xorg.conf files, 33–39
- Ubuntu Budgie, 49
- Ubuntu Core, 103
- Ubuntu Kylin, 50
- Ubuntu MATE, 48–49
- Ubuntu QA Teams, 646–647
- Ubuntu Software, 91
- Ubuntu Testing Teams, 646
- ubuntu-restricted-extras, 76
- UEFI
 - boot process, 273–274
 - Ubuntu installations, 6
- ufw command, 369
- UFW, configuring, 386–387
- UID, 204, 448
- unexpanded variables, maintaining, 249
- unicast addressing, 338
- uninstalling Nginx web server, 470
- Unison, 315
- units (services), listing, 276
- UNIX
 - backup levels, 306
 - CUPS GUI, network printers, 431–434
 - sound, 63–64
- UnQL, 511
- until statements, 263
- updating
 - software, 12–14
 - Software Updater, 94–95
 - Ubuntu
 - first updates, 11
 - installations, downloading updates during installation, 8
 - Software Updater, 12–14
- uplink ports, 342
- uptime command, 286
- usage statistics, user accounts, 212
- USB
 - digital cameras, 70
 - thumb drives, Ubuntu installations, 3, 6–11
- used memory, displaying, 286
- user accounts, 201–202, 204, 207
 - adding accounts, 209–211

- administration privileges, 217
- BOFH, 205
- changing identities, 217–219
- creating, 136–137
- deleting, 137
- disk quotas, managing, 222–223
- file permissions, 204
- GID, 204
- group management, 205
 - finding groups, 206
 - listing groups, 205–206
 - tools, 206–207
- locking users out of accounts, 209
- lusers, 205
- monitoring user activity, 211–212
- online references, 224
- passwords, 202, 212, 216
 - changing in batches, 216–217
 - password files, 212–214
 - shadow passwords, 214–215
 - system password policies, 212
- privileges, 109–111
- root privileges, 219–222
- root user accounts, 202–203
- stereotypes, 205
- substitute user accounts, 217–218
- super user accounts, 109–111, 133–134, 202–203, 217–218
- system user accounts, 203–204
- tools, 208–209
- Ubuntu commands, 223–224
- UID, 204
- usernames, 211
- User directive (Apache web server), 448
- user directories, 115
- user variables, 242
- useradd command, 208, 224

- useradd -D command, 208
- useradd -G command, 207
- UserDir directive (Apache web server), 449
- usermod command, 209, 224
- usermod -G command, 207
- users, productivity suites, 56
- /usr directory
 - shared data, 117
 - X server, 33
- UTP cabling, 341

V

- Vala, 633
- values, assigning to variables, 242
- /var directory, 117
- variable data files, /var directory, 117
- variables
 - built-in variables, 242, 246–247
 - environment variables, 182–185, 242
 - shell scripts, 242
 - accessing values, 243
 - accessing/retrieving variables, 244
 - assigning values, 242
 - resolving strings with embedded spaces, 248–249
 - unexpanded variables, maintaining, 249
 - user variables, 242
- VBA, LibreOffice and Office compatibility, 55
- version control
 - configuration files, 320–322
 - systems, 609–610
- vi text editor, 185, 187–188
- video
 - .avi files, 75
 - dvd+rw-tools, 74
 - DVD/CD burning applications

- Brasero, 71
- Linux command line, CD creation, 72–73
- Linux command line, DVD creation, 73–75
- Ubuntu support, 71
- editing, 76, 77
 - Avidemux, 77
 - Blender, 77
 - Cinelerra, 77
 - DaVinci Resolve, 77
 - Kdenlive, 77
 - Lightworks, 77
 - OpenShot Video Editor, 77
 - PiTiVi, 77
 - Shotcut, 77
- .flv files, 75
- .mov files, 75
- .mpeg files, 75
- .ogv/.ogg files, 75
- .qt files, 75
- Totem Movie Player, 76
- ubuntu-restricted-extras, 76
- video drivers (proprietary), installing, 80–81
- viewing
 - in Linux, 76
 - video formats, 75
- VLC, 76
 - .webm files, 75
- viewing
 - command history, 197
 - default file permissions, 124–125
 - video
 - in Linux, 76
 - video formats, 75
- vim text editor, 185
- vimtutor command, 188
- virt-clone, 526
- virt-install, 525–526
- virtual hosting
 - Apache web server, 461–462
 - Nginx web server, 473–474
- VirtualBox, 527–528
- virtualization, 521–523
 - bridged networks, 524–525
 - KVM, 523–526
 - online references, 529
 - VirtualBox, 527–528
 - VM, 522, 525–526, 536
 - connections, 526
 - copying, 526
 - starting, 526
 - stopping, 526
 - virt-clone, 526
 - virt-install, 525–526
 - vmbuilder, 525
 - VMware, 528
 - Xen, 528
- virus scanners, email, 562
- viruses, 385
- Visual Studio Code, 606
- visudo command, 135–136
- VLC, 76
- VM, 522, 536
 - connections, 526
 - copying, 526
 - KVM, 523–526
 - starting, 526
 - stopping, 526
 - virt-clone, 526
 - vmbuilder, 525
- vmstat command, 286
- VMware, 528
- VNC, 375–377
- vncviewer, 297

VPN, 563, 572–573

 configuring

 clients, 573–575

 servers, 575–577

 servers, online references, 577

vulnerability assessments, 381–382

W

waking computers from sleep, 231–233

war driving, 383

Warsow, 82

.wav files, 65

Wayland, 32

web browsers

 Chrome, 53–54

 Chromium, 53–54

 database access, 503

 Firefox, 52–53

 online references, 54

.webm files, 75

web resources

 Apache web server, 466

 backups, 324

 BIND, 595

 boot process, 278–279

 cloud computing, 541

 command line, 139, 165, 200

 containers, 534

 desktop environments, 50

 DNS, 595

 file sharing, 435

 games, 89

 HTTP servers, 482

 kernels, 419–420

 LDAP, 585

 Linux, 23, 28

 multimedia applications, 77

 MySQL, 493, 507

 network printers, 435

 networks, 369–370

 Nginx web server, 477

 NoSQL databases, 519–520

 performance tuning, 396–400

 PostgreSQL, 493, 507

 programming, 618–619

 programming languages, 633–634

 QA, 647

 remote access, 377–378

 security, 391–392

 sets of servers, 545

 shell scripts, 270

 software management, 104

 sound, 65

 Squid, 577

 system monitoring, 299

 system resource management, 104

 testing, 647

 Ubuntu, 22, 28, 30

 Ubuntu development, 643

 user accounts, 224

 virtualization, 529

 VPN, 577

 Wayland, 32

 web browsers, 54, 61

 web server stacks, 441

 X server, 40

web server stacks, 437

 LAMP, 437–439

 LEMP, 439

 MEAN, 437

 online references, 441

web servers

Apache web server

- access control, 452, 455
- configuring, 446–449, 450–452
- development of, 443–444
- file system authentication, 452–455
- .htaccess configuration files, 450–452, 469
- HTTPS, 464–466
- installing, 444
- logging, 463–464
- modules, 455–461
- MPM, 449–450
- online references, 466
- overview of, 444
- runtime servers, 446–449
- security, 444, 452–455
- starting/stopping, 444–446
- threads, 468
- virtual hosting, 461–462

C10K problem, 467–468

event handlers, 468–469

Nginx web server, 467–469

- building sources, 469–470
- configuring, 470–473
- HTTPS, 476–477
- installing, 469–470
- modules, 475
- online references, 477
- PHP, 474–475
- uninstalling, 470
- virtual hosting, 473–474

threads, 468

wget command, 164

whatis command, 160

whereis command, 112, 160

which command, 164

while statements, 261, 262

wide column stores, 517

widgets, 42

wildcard characters. *See also* symbols

- commands and, 235
- directories, 133

Wildfly, 482

window managers

- changing, 39
- X server, 33–34

Windows

- CrossOver Office, 61
- dual-boot systems, 6
- games, 88–89
- productivity applications, 61
- Wine, 61

Wine, 61

wireless networks, 361

- configuring, 20
- interfaces, 340
- security, 382–383
- selecting protocols, 363
- Ubuntu support, 361–363

Wireshark, 298, 330

worms, 380

writing, shell scripts, 237–239

WYGIWYW, 60

WYSIWYG, 60

X

X server, 32, 33

- benefits of, 33–34
- components of, 33–34
- display manager, 39
- display managers, 33

- distributed processing, 32–33
- online references, 40
- starting, 39
- terminal clients, 33–34
- /usr directory, 33
- window manager, 33–34
- xorg.conf files, 33–34
 - components of, 34–35
 - Device section, 37–38
 - Files section, 35–36
 - InputDevice section, 36–37
 - Module section, 36
 - Monitor section, 37
 - Screen section, 38–39
 - ServerLayout section, 35
- Xara Xtreme, 70
- xconfig command, 414–417
- xDSL Internet access, 365
- Xen, 528
- Xfce, 46
- XML, 59
 - DocBook, Publican, 59–60
 - gedit, 59
 - Publican, 59–60
 - XML Copy Editor, 60

- XML Copy Editor, 60
- X.Org, 33
- xorg.conf files, 33–34
 - components of, 34–35
 - Device section, 37–38
 - Files section, 35–36
 - InputDevice section, 36–37
 - Module section, 36
 - Monitor section, 37
 - Screen section, 38–39
 - ServerLayout section, 35
- Xubuntu, 46–47
- xz command, 192

Y

- YAML, /etc/netplan/*.yaml files, 353–355
- Yaws, 480

Z

- Zabbix, 299
- zsh, 226

BONUS CHAPTER 42

Using Perl

Perl (whose name comes from the *Practical Extraction and Report Language* or the *Pathologically Eclectic Rubbish Lister*, depending on whom you speak to) is a powerful scripting tool that enables you to manage files, create reports, edit text, and perform many other tasks. Perl is included with and installed in Ubuntu by default and could be considered an integral part of the distribution because Ubuntu depends on Perl for many types of software services, logging activities, and software tools.

Perl is not the easiest of programming languages to learn because it is designed for flexibility. This chapter shows how to create and use Perl scripts on your system. You learn what a Perl program looks like, how the language is structured, and where you can find modules of prewritten code to help you write your own Perl scripts. This chapter also includes several examples of Perl used to perform a few common functions on a computer system.

Using Perl with Linux

Although originally designed as a data-extraction and report-generation language, Perl appeals to many Linux system administrators because they can use it to create utilities that fill a gap between the capabilities of shell scripts and compiled C programs. Another advantage of Perl over other UNIX tools is that it can process and extract data from binary files, whereas `sed` and `awk` cannot.

NOTE

In Perl, “there is more than one way to do it.” This is the unofficial motto of Perl, and it comes up so often that it is usually abbreviated as TIMTOWTDI.

IN THIS CHAPTER

- ▶ Using Perl with Linux
- ▶ Perl Variables and Data Structures
- ▶ Perl Operators
- ▶ Conditional Statements: `if/else` and `unless`
- ▶ Looping
- ▶ Regular Expressions
- ▶ Access to the Shell
- ▶ Modules and CPAN
- ▶ Code Examples
- ▶ References

You can use Perl at your shell's command line to execute one-line Perl programs, but most often the programs (usually ending in `.pl`) are run as commands. These programs generally work on any computer platform because Perl has been ported to nearly every operating system.

Perl programs are used to support a number of Ubuntu services, such as system logging. For example, if you install the `logwatch` package, the `logwatch.pl` program is run every morning at 6:25 a.m. by the `crond` (scheduling) daemon on your system. Other Ubuntu services supported by Perl include the following:

- ▶ Amanda for local and network backups
- ▶ Fax spooling with the `faxrunqd` program
- ▶ Printing supported by Perl document-filtering programs
- ▶ Hardware sensor monitoring setup using the `sensors-detect` Perl program

Perl Versions

Perl is installed in Ubuntu by default. You can download the code from www.perl.com and build the newest version from source if you want to, although a stable and quality release of Perl is already installed by default in Ubuntu and most (perhaps all) Linux and UNIX-like distributions, including macOS. Updated versions might appear in the Ubuntu repositories, but they're generally only security fixes that can be installed by updating your system. See Chapter 9, "Managing Software," to see how to quickly get a list of available updates for Ubuntu.

You can determine what version of Perl you have installed by typing `perl -v` at a shell prompt. When you install the latest Ubuntu distribution, you should have the latest version of Perl that was available when the software for your Ubuntu release was gathered and finalized.

Note that there was a development effort underway to replace Perl 5. The new version was to be called Perl 6, but had diverged far enough away from Perl 5 that the decision was made to rename it to Raku. You can learn about Raku at www.raku.org.

A Simple Perl Program

This section introduces a very simple Perl program example to get you started using Perl. Although trivial for experienced Perl hackers, this short example is necessary for new users who want to learn more about Perl.

To introduce you to the absolute basics of Perl programming, Listing 42.1 illustrates a simple Perl program that prints a short message.

LISTING 42.1 A Simple Perl Program

```
#!/usr/bin/perl
print 'Look at all the camels!\n';
```

Type in the program shown in the listing and save it to a file called `trivial.pl`. Then make the file executable by using the `chmod` command (see the following sidebar) and run it at the command prompt.

Command-Line Error

If you get the message `bash: trivial.pl: command not found` or `bash: ./trivial.pl: Permission denied`, you have either typed the command line incorrectly or forgotten to make `trivial.pl` executable with the `chmod` command, as shown here:

```
matthew@seymour:~$ chmod +x trivial.pl
```

You can force the command to execute in the current directory as follows:

```
matthew@seymour:~$ ./trivial.pl
```

Or you can use Perl to run the program, like this:

```
matthew@seymour:~$ perl trivial.pl
```

The sample program in the listing is a two-line Perl program. When you type in the program and run it (using Perl or by making the program executable), you are creating a Perl program, a process duplicated by Linux users around the world every day.

NOTE

`#!` is often pronounced *shebang*, which is short for *sharp* (musicians' name for the `#` character), and *bang*, which is another name for the exclamation point. This notation is also used in shell scripts. See Chapter 14, "Automating Tasks and Shell Scripting," for more information about writing shell scripts.

The `#!` line is technically not part of the Perl code at all. The `#` character indicates that the rest of the screen line is a comment. The comment is a message to the shell, telling it where it should go to find the executable to run this program. The interpreter ignores the comment line. Comments are useful for documenting scripts, like this:

```
#!/usr/bin/perl
# a simple example to print a greeting
print "hello there\n";
```

The `#` character can also be used in a quoted string and used as the delimiter in a regular expression.

A block of code such as what might appear inside a loop or a branch of a conditional statement is indicated with curly braces (`{}`). For example, here is an infinite loop:

```
#!/usr/bin/perl
# a block of code to print a greeting forever
while (1) {
```

```
print "hello there\n";
};
```

A Perl statement is terminated with a semicolon (;). A Perl statement can extend over several screen lines because Perl is not concerned about white space.

The second line of the simple program in Listing 42.1 prints the text enclosed in quotation marks. `\n` is the escape sequence for a newline character.

TIP

Using the `perldoc` and `man` commands is an easy way to get more information about the version of Perl installed on your system. To learn how to use the `perldoc` command, enter the following:

```
matthew@seymour:~$ perldoc
```

To get introductory information on Perl, you can use either of these commands:

```
matthew@seymour:~$ perldoc perl
```

```
matthew@seymour:~$ man perl
```

For an overview or table of contents of Perl's documentation, use the `perldoc` command, like this:

```
matthew@seymour:~$ perldoc perltoc
```

The documentation is extensive and well organized. Perl includes a number of standard Linux manual pages as brief guides to its capabilities, but perhaps the best way to learn more about Perl is to read its `perlfunc` document, which lists all the available Perl functions and their usage. You can view this document by using the `perldoc` script and typing `perldoc perlfunc` at the command line. You can also find this document online at <https://perldoc.perl.org>.

Perl Variables and Data Structures

Perl is a *weakly typed* language, meaning that it does not require that you declare a data type, such as a type of value (data) to be stored in a particular variable. C, for example, makes you declare that a particular variable is an integer, a character, a structure, or whatever the case may be. Perl variables are whatever type they need to be and can change type when you need them to.

Perl Variable Types

Perl has three variable types: *scalars*, *arrays*, and *hashes*. A different character is used to signify each variable type, so you can have the same name used with each type at the same time.

A scalar variable is indicated with the `$` character, as in `$penguin`. Scalars can be numbers or strings, and they can change type as needed. If you treat a number like a string, it becomes a string. If you treat a string like a number, it is translated into a number if it makes sense to do so; otherwise, it usually evaluates to `0`. For example, the string `"76trombones"` evaluates as the number `76` if used in a numeric calculation, but the string `"polar bear"` evaluates to `0`.

A Perl array is indicated with the `@` character, as in `@fish`. An *array* is a list of values referenced by index number, starting with the first element, numbered `0`, just as in `C` and `awk`. Each element in the array is a scalar value. Because scalar values are indicated with the `$` character, a single element in an array is also indicated with a `$` character.

For example, `$fish[2]` refers to the third element in the `@fish` array. This tends to throw some people off but is similar to arrays in `C`, in which the first array element is `0`.

A hash is indicated with the `%` character, as in `%employee`. A *hash* is a list of name/value pairs. Individual elements in a hash are referenced by name rather than by index (unlike in an array). Again, because the values are scalars, the `$` character is used for individual elements.

For example, `$employee{name}` gives you one value from the hash. Two rather useful functions for dealing with hashes are `keys` and `values`. The `keys` function returns an array that contains all the keys of the hash, and `values` returns an array of the values of the hash. Using this approach, the Perl program in Listing 42.2 displays all the values in your environment, much like typing the `bash` shell's `env` command.

LISTING 42.2 Displaying the Contents of the `env` Hash

```
#!/usr/bin/perl
foreach $key (keys %ENV) {
    print "$key = $ENV{$key}\n";
}
```

Special Variables

Perl has a variety of special variables, which usually look like punctuation—`$_`, `#!`, and `$]`—and are all extremely useful for shorthand code. `$_` is the default variable, `#!` is the error message returned by the operating system, and `$]` is the Perl version number.

`$_` is perhaps the most useful of these. You see this variable used often in this chapter. `$_` is the Perl default variable, which is used when no argument is specified. For example, the following two statements are equivalent:

```
chomp;
```

```
chomp($_);
```

The following loops are equivalent:

```
for $cow (@cattle) {
```

```

    print "$cow says moo.\n";
}

for (@cattle) {
    print "$_ says moo.\n";
}

```

For a complete list of the special variables, see the `perlvar` man page.

Perl Operators

Perl supports a number of operators for performing various operations. There are *comparison* operators (used to compare values, as the name implies), *compound* operators (used to combine operations or multiple comparisons), *arithmetic* operators (to perform math), and special string constants.

Comparison Operators

The comparison operators used by Perl are similar to those used by C, `awk`, and the `cs` shells, and they are used to specify and compare values (including strings). A comparison operator is most often used within an `if` statement or loop. Perl has comparison operators for numbers and strings. Table 42.1 shows the numeric comparison operators and their meanings.

Table 42.1 Numeric Comparison Operators in Perl

Operator	Meaning
<code>==</code>	Is equal to
<code><</code>	Less than
<code>></code>	Greater than
<code><=</code>	Less than or equal to
<code>>=</code>	Greater than or equal to
<code><=></code>	Returns <code>-1</code> if less than, <code>0</code> if equal to, and <code>1</code> if greater than
<code>!=</code>	Not equal to
<code>..</code>	Range of <code>>=</code> first operand to <code><=</code> second operand

Table 42.2 shows the string comparison operators and their meanings.

Table 42.2 String Comparison Operators in Perl

Operator	Meaning
<code>eq</code>	Is equal to
<code>lt</code>	Less than
<code>gt</code>	Greater than
<code>le</code>	Less than or equal to
<code>ge</code>	Greater than or equal to
<code>ne</code>	Not equal to

Operator	Meaning
<code>cmp</code>	Returns -1 if less than, 0 if equal to, and 1 if greater than
<code> =~ </code>	Matched by regular expression
<code> !~ </code>	Not matched by regular expression

Compound Operators

Perl uses compound operators, similar to those used by C or `awk`, which can be used to combine other operations (such as comparisons or arithmetic) into more complex forms of logic. Table 42.3 shows the compound pattern operators and their meanings.

Table 42.3 Compound Pattern Operators in Perl

Operator	Meaning
<code>&&</code>	Logical AND
<code> </code>	Logical OR
<code>!</code>	Logical NOT
<code>()</code>	Parentheses; used to group compound statements

Arithmetic Operators

Perl supports a variety of math operations. Table 42.4 summarizes these operators.

Table 42.4 Perl Arithmetic Operators

Operator	Purpose
<code>x**y</code>	Raises x to the y power (same as x^y)
<code>x%y</code>	Calculates the remainder of x/y
<code>x+y</code>	Adds x to y
<code>x-y</code>	Subtracts y from x
<code>x*y</code>	Multiplies x by y
<code>x/y</code>	Divides x by y
<code>-y</code>	Negates y (switches the sign of y); also known as the unary minus
<code>++y</code>	Increments y by 1 and uses the value (prefix increment)
<code>y++</code>	Uses the value of y and then increments by 1 (postfix increment)
<code>--y</code>	Decrements y by 1 and uses the value (prefix decrement)
<code>y--</code>	Uses the value of y and then decrements by 1 (postfix decrement)
<code>x=y</code>	Assigns the value of y to x . Perl also supports operator-assignment operators (<code>+=</code> , <code>-=</code> , <code>*=</code> , <code>/=</code> , <code>%=</code> , <code>**=</code> , and others)

You can also use comparison operators (such as `==` or `<`) and compound pattern operators (`&&`, `||`, and `!`) in arithmetic statements. They evaluate to the value 0 for false and 1 for true.

Other Operators

Perl supports a number of operators that do not fit any of the prior categories. Table 42.5 summarizes these operators.

Table 42.5 Other Perl Operators

Operator	Purpose
<code>~x</code>	Bitwise NOT (changes 0 bits to 1 and 1 bits to 0 bits)
<code>x & y</code>	Bitwise AND
<code>x y</code>	Bitwise OR
<code>x ^ y</code>	Bitwise exclusive or (XOR)
<code>x << y</code>	Bitwise shift left (shifts <code>x</code> by <code>y</code> bits)
<code>x >> y</code>	Bitwise shift right (shifts <code>x</code> by <code>y</code> bits)
<code>x . y</code>	Concatenate <code>y</code> onto <code>x</code>
<code>a x b</code>	Repeats string <code>a</code> for <code>b</code> number of times
<code>x, y</code>	Comma operator; evaluates <code>x</code> and then <code>y</code>
<code>x ? y : z</code>	Conditional expression (If <code>x</code> is true, <code>y</code> is evaluated; otherwise, <code>z</code> is evaluated.)

Except for the comma operator and conditional expression, you can also use these operators with the assignment operator, similar to the way addition (+) can be combined with assignment (=), giving +=.

Special String Constants

Perl supports string constants that have special meaning or cannot be entered from the keyboard.

Table 42.6 shows most of the constants supported by Perl.

Table 42.6 Perl Special String Constants

Expression	Meaning
<code>\\</code>	The means of including a backslash
<code>\a</code>	The alert or bell character
<code>\b</code>	Backspace
<code>\cC</code>	Control character (like holding the Ctrl key down and pressing the C character)
<code>\e</code>	Escape
<code>\f</code>	Formfeed
<code>\n</code>	Newline
<code>\r</code>	Carriage return
<code>\t</code>	Tab
<code>\xNN</code>	Indicates that <code>NN</code> is a hexadecimal number
<code>\oNNN</code>	Indicates that <code>NNN</code> is an octal (base 8) number

Conditional Statements: `if/else` and `unless`

Perl offers two conditional statements, `if` and `unless`, which function opposite one another. `if` enables you to execute a block of code only if certain conditions are met so that you can control the flow of logic through your program. Conversely, `unless` performs the statements when certain conditions are not met.

The following sections explain and demonstrate how to use these conditional statements when writing scripts for Linux.

`if`

The syntax of the Perl `if/else` structure is as follows:

```
if (condition) {  
    statement or block of code  
} elsif (condition) {  
    statement or block of code  
} else {  
    statement or block of code  
}
```

condition is a statement that returns a true or false value.

Truth is defined in Perl in a way that might be unfamiliar to you, so be careful. Everything in Perl is true except `0` (the digit zero), `"0"` (the string containing the number 0), `"` (the empty string), and an undefined value. Note that even the string `"00"` is a true value because it is not one of the four false cases.

A *statement or block of code* section is executed if the test condition returns a true value.

For example, Listing 42.3 uses the `if/else` structure and shows conditional statements using the `eq` string comparison operator.

LISTING 42.3 `if/elsif/else`

```
if ($favorite eq "chocolate") {  
    print "I like chocolate too.\n";  
} elsif ($favorite eq "spinach") {  
    print "Oh, I do not like spinach.\n";  
} else {  
    print "Your favorite food is $favorite.\n";  
}
```

`unless`

`unless` works just like `if` only backward. `unless` performs a statement or block if a condition is false:

```
unless ($name eq "Rich") {
```

```
print "Go away, you're not allowed in here!\n";
}
```

NOTE

You can restate the preceding example in more natural language, like this:

```
print "Go away!\n" unless $name eq "Rich";
```

Looping

A *loop* repeats a program action multiple times. A simple example is a countdown timer that performs a task (waiting for one second) 300 times before telling you that your egg is done boiling.

Looping constructs (also known as *control structures*) can be used to iterate a block of code as long as certain conditions apply or while the code steps through (evaluates) a list of values, perhaps using that list as arguments.

Perl has several looping constructs.

for

The `for` construct performs a *statement* (block of code) for a set of conditions defined as follows:

```
for (start condition; end condition; increment function) {
    statement(s)
}
```

The `start condition` is set at the beginning of the loop. Each time the loop is executed, the `increment function` is performed until the end condition is achieved. This looks much like the traditional `for/next` loop. The following code is an example of a `for` loop:

```
for ($i=1; $i<=10; $i++) {
    print "$i\n"
}
```

foreach

The `foreach` construct performs a statement block for each element in a list or an array:

```
@names = ("alpha","bravo","Charlie");
foreach $name (@names) {
    print "$name sounding off!\n";
}
```

The loop variable (`$name` in the example) is not merely set to the value of the array elements; it is aliased to that element. This means if you modify the loop variable, you're

actually modifying the array. If no loop array is specified, the Perl default variable `$_` may be used, as shown here:

```
@names = ("alpha", "bravo", "Charlie");
foreach (@names) {
    print "$_ sounding off!\n";
}
```

This syntax can be very convenient, but it can also lead to unreadable code. Give a thought to the poor person who'll be maintaining your code. (It will probably be you.)

NOTE

`foreach` is frequently abbreviated as `for`.

while

`while` performs a block of statements as long as a particular condition is true, as shown in this example:

```
while ($x<10) {
    print "$x\n";
    $x++;
}
```

Remember that the condition can be anything that returns a true or false value. For example, it could be a function call, like this:

```
while ( InvalidPassword($user, $password) ) {
    print "You've entered an invalid password. Please try again.\n";
    $password = GetPassword;
}
```

until

`until` is exactly the opposite of the `while` statement. It performs a block of statements as long as a particular condition is false (or, rather, until it becomes true). The following is an example:

```
until (ValidPassword($user, $password)) {
    print "YSdpgm_m
Sdpgm_m
You have entered an invalid password. Please try again.\n";
Sdpgm_m
    $password = GetPassword;
}
```

last and next

You can force Perl to end a loop early by using a `last` statement. `last` is similar to the C `break` command; the loop is exited. If you decide you need to skip the remaining contents of a loop without ending the loop, you can use `next`, which is similar to the C `continue` command. Unfortunately, these statements do not work with `do ... while`. However, you can use `redo` to jump to a loop (marked by a label) or inside the loop where called:

```
$a = 100;
while (1) {
    print "start\n";
    TEST: {
        if (($a = $a / 2) > 2) {
            print "$a\n";
            if (-$a < 2) {
                exit;
            }
            redo TEST;
        }
    }
}
```

In this simple example, the variable `$a` is repeatedly manipulated and tested in a loop. The word `start` will be printed only once.

do ... while and do ... until

The `while` and `until` loops evaluate the conditional first. The behavior is changed by applying a `do` block before the conditional. With the `do` block, the condition is evaluated last, which results in the contents of the block always executing at least once (even if the condition is false). This is similar to the C language `do ... while (conditional)` statement.

Regular Expressions

Perl's greatest strength lies in text and file manipulation, which is accomplished by using the *regular expression* (*regex*) library. Regexes, which are quite different from the wildcard-handling and filename-expansion capabilities of the shell (see Chapter 14, "Automating Tasks and Shell Scripting"), allow complicated pattern matching and replacement to be done efficiently and easily.

For example, the following line of code replaces every occurrence of the string `bob` or the string `mary` with `fred` in a line of text:

```
$string =~ s/bob|mary/fred/gi;
```

Without going into too many of the details, Table 42.7 explains what the preceding line says.

Table 42.7 Explanation of `$string =~ s/bob|mary/fred/gi;`

Element	Explanation
<code>\$string =~</code>	Performs this pattern match on the text found in the variable called <code>\$string</code> .
<code>s</code>	Performs a substitution.
<code>/</code>	Begins the text to be matched.
<code>bob mary</code>	Matches the text <code>bob</code> or <code>mary</code> . You should remember that it is looking for the text <code>mary</code> , not the word <code>mary</code> ; that is, it will also match the text <code>mary</code> in the word <code>maryland</code> .
<code>fred</code>	Replaces anything that was matched with the text <code>fred</code> .
<code>/</code>	Ends replace text.
<code>g</code>	Does this substitution globally; that is, replaces the match text wherever in the string you match it (and any number of times).
<code>i</code>	The search text is not case sensitive. It matches <code>bob</code> , <code>Bob</code> , or <code>bOB</code> .
<code>;</code>	Indicates the end of the line of code.

If you are interested in the details, you can get more information using the `regex (7)` section of the man page by entering `man 7 regex` from the command line.

Although replacing one string with another might seem a rather trivial task, the code required to do the same thing in another language (for example, C) is rather daunting unless supported by additional subroutines from external libraries.

Access to the Shell

Perl can perform for you any process you might ordinarily perform by typing commands to the shell through the `\`` syntax. For example, the code in Listing 42.4 prints a directory listing.

LISTING 42.4 Using Backticks to Access the Shell

```
$curr_dir = `pwd`;
@listing = `ls -al`;
print "Listing for $curr_dir\n";
foreach $file (@listing) {
    print "$file";
}
```

NOTE

The `\`` notation uses the backtick found above the Tab key (on most keyboards), not the single quotation mark.

You can also use the `Shell` module to access the shell. `Shell` is one of the standard modules that come with Perl; it allows creation and use of a shell-like command line. The following code provides an example:

```
use Shell qw(cp);
cp ("/home/httpd/logs/access.log", :/tmp/httpd.log");
```

This code almost looks like it is importing the command-line functions directly into Perl. Although that is not really happening, you can pretend that the code is similar to a command line and use this approach in your Perl programs.

A third method of accessing the shell is via the `system` function call:

```
$rc = 0xffff & system(`cp /home/httpd/logs/access.log /tmp/httpd.log`);
if ($rc == 0) {
    print "system cp succeeded \n";
} else {
    print "system cp failed $rc\n";
}
```

The call can also be used with the `or die` clause:

```
system(`cp /home/httpd/logs/access.log /tmp/httpd.log`) == 0
    or die "system cp failed: $?"
```

However, you cannot capture the output of a command executed through the `system` function.

Modules and CPAN

A great strength of the Perl community (and the Linux community) is the fact that it is an open source community. This community support is expressed for Perl via the *Comprehensive Perl Archive Network (CPAN)*, which is a network of mirrors of a repository of Perl code.

Most of CPAN is made up of *modules*, which are reusable chunks of code that do useful things, similar to software libraries containing functions for C programmers. These modules help speed development when building Perl programs and free Perl hackers from repeatedly reinventing the wheel when building bicycles.

Perl comes with a set of standard modules installed. Those modules should contain much of the functionality that you initially need with Perl. If you need to use a module not installed with Ubuntu, use the CPAN module (which is one of the standard modules) to download and install other modules onto your system. At <https://cpan.perl.org>, you can find the CPAN Multiplex Dispatcher, which attempts to direct you to the CPAN site closest to you.

Typing the following command puts you into an interactive shell that gives you access to CPAN. You can type `help` at the prompt to get more information on how to use the CPAN program:

```
matthew@seymour:~$ perl -MCPAN -e shell
```

After installing a module from CPAN (or writing one of your own), you can load that module into memory, where you can use it with the `use` function:

```
use Time::CTime;
```

`use` looks in the directories listed in the variable `@INC` for the module. In this example, `use` looks for a directory called `Time`, which contains a file called `CTime.pm`, which in turn is assumed to contain a package called `Time::CTime`. The distribution of each module should contain documentation on using that module.

For a list of all the standard Perl modules (those that come with Perl when you install it), see `perlmodlib` in the Perl documentation. You can read this document by typing `perldoc perlmodlib` at the command prompt.

Code Examples

The following sections contain a few examples of things you might want to do with Perl.

Sending Mail

You can get Perl to send email in several ways. One method that you see frequently is opening a pipe to the `sendmail` command and sending data to it (as shown in Listing 42.5). Another method is using the `Mail::Sendmail` module (available through CPAN), which uses socket connections directly to send mail (as shown in Listing 42.6). The latter method is faster because it does not have to launch an external process. Note that `sendmail` must be installed on your system for the Perl program in Listing 42.5 to work.

LISTING 42.5 Sending Mail Using `Sendmail`

```
#!/usr/bin/perl
open (MAIL, "| /usr/sbin/sendmail -t"); # Use -t to protect from users
print MAIL <<EndMail;
To: you\
From: me\
Subject: A Sample Email\nSending email from Perl is easy!\n
.
EndMail
close MAIL;
```

NOTE

The `@` sign in the email addresses must be escaped so that Perl does not try to evaluate an array of that name. That is, `dpitts@mk.net` will cause a problem, so you need to use this:

```
dpitts\<indexterm startref="iddle2799" class="endofrange"
significance="normal"/>:}}
```


The syntax used to print the mail message is called a *here document*. The syntax is as follows:

```
print <<EndText;
.....
EndText
```

The *EndText* value must be identical at the beginning and at the end of the block, including any white space.

LISTING 42.6 Sending Mail Using the Mail::Sendmail Module

```
#!/usr/bin/perl
use Mail::Sendmail;

%mail = ( To => "you@there.com",
          From => "me@here.com",
          Subject => "A Sample Email",
          Message => "This is a very short message"
        );

sendmail(%mail) or die $Mail::Sendmail::error;

print "OK. Log says:\n", $Mail::Sendmail::log;

use Mail::Sendmail;
```

Perl ignores the comma after the last element in the hash. It is convenient to leave it there; then, if you want to add items to the hash, you do not need to add the comma. This is purely a style decision.

Using Perl to Install a CPAN Module

You can use Perl to interactively download and install a Perl module from the CPAN archives by using the `-M` and `-e` commands. Start the process by using a Perl command like this:

```
# perl -MCPAN -e shell
```

When you press Enter, you see some introductory information, and you are asked to choose an initial automatic or manual configuration, which is required before any download or install takes place. Type `no` and press Enter to have Perl automatically configure for the download and install process; or if you want, just press Enter to manually configure for downloading and installation. If you use manual configuration, you must answer a series of questions regarding paths, caching, terminal settings, program locations, and so on. Settings are saved in a directory named `.cpan` in the current directory.

When finished, you see the CPAN prompt:

```
cpan>
```

To have Perl examine your system and then download and install a large number of modules, use the `install` keyword, specify `Bundle` at the prompt, and then press Enter, like this:

```
cpan> install Bundle::CPAN
```

To download a desired module (using the example in Listing 42.6), use the `get` keyword, like this:

```
cpan> get Mail::Sendmail
```

The source for the module is downloaded into the `.cpan` directory. You can then build and install the module by using the `install` keyword, like this:

```
cpan> install Mail::Sendmail
```

The entire process of retrieving, building, and installing a module can also be accomplished at the command line by using Perl's `-e` option, like this:

```
# perl -MCPAN -e "install Mail::Sendmail"
```

Note also that the `@` sign in Listing 42.6 does not need to be escaped within single quotation marks (`' '`). Perl does not *interpolate* (evaluate variables) within single quotation marks but does within double quotation marks and here strings (similar to `<<` shell operations).

Purging Logs

Many programs maintain some variety of logs. Often, much of the information in the logs is redundant or just useless. The program shown in Listing 42.7 removes all lines from a file that contain a particular word or phrase, so lines that you know are not important can be purged. For example, you might want to remove all the lines in the Apache error log that originate with your test client machine because you know those error messages were produced during testing.

LISTING 42.7 Purging Log Files

```
#!/usr/bin/perl
# Be careful using this program!
# This will remove all lines that contain a given word
# Usage:  remove <word> <file>
$word=@ARGV[0];
$file=@ARGV[1];
if ($file) {
    # Open file for reading
    open (FILE, "$file") or die "Could not open file: $!";
    @lines=<FILE>;
    close FILE;
    # Open file for writing
```

```

open (FILE, ">$file") or die "Could not open file for writing: $!";
for (@lines) {
    print FILE unless /$word/;
} # End for
close FILE;
} else {
    print "Usage:  remove <word> <file>\n";
} # End if...else

```

The code in Listing 42.7 includes a few idiomatic Perl expressions to keep it brief. It reads the file into an array by using the `<FILE>` notation; it then writes the lines back out to the file unless they match the pattern given on the command line.

The `die` function kills program operation and displays an error message if the `open` statements fail. `$!` in the error message, as mentioned earlier in this chapter, is the error message returned by the operating system. It is likely to be something like 'file not found' or 'permission denied'.

Posting to Usenet

If some portion of your job requires periodic postings to Usenet—an FAQ listing, for example—the following Perl program can automate the process for you. In Listing 42.8, the posted text is read in from a text file, but your input can come from anywhere.

The program shown in Listing 42.8 uses the `Net::NNTP` module, which is a standard part of the Perl distribution. You can find more documentation on the `Net::NNTP` module by entering 'perldoc Net::NNTP' at the command line.

LISTING 42.8 Posting an Article to Usenet

```

#!/usr/bin/perl
# load the post data into @post
open (POST, "post.file");
@post = <POST>;
close POST;
# import the NNTP module
use Net::NNTP;
$NNTPhost = 'news';
# attempt to connect to the remote host;
# print an error message on failure
$nnntp = Net::NNTP->new($NNTPhost)
    or die "Cannot contact $NNTPhost: $!";
# $nnntp->debug(1);
$nnntp->post()
    or die "Could not post article: $!";
# send the header of the post
$nnntp->datasend("Newsgroups: news.announce\n");
$nnntp->datasend("Subject: FAQ - Frequently Asked Questions\n");

```

```
$nntp->datasend("From: ADMIN <root>\n")
$nntp->datasend("\n\n");
# for each line in the @post array, send it
for (@post) {
    $nntp->datasend($_);
} # End for
$nntp->quit;
```

One-Liners

Perl excels at the one-liner. Folks go to great lengths to reduce tasks to one line of Perl code. Perl has the rather undeserved reputation of being unreadable. The fact is that you can write unreadable code in any language. Perl allows for more than one way to do something, and this leads rather naturally to people trying to find the most arcane way to do things.

Named for Randal Schwartz, a *Schwartzian* transform is a way of sorting an array by something that is not obvious. The `sort` function sorts arrays alphabetically; that is pretty obvious. What if you want to sort an array of strings alphabetically by the third word? Perhaps you want something more useful, such as sorting a list of files by file size? A Schwartzian transform creates a new list that contains the information you want to sort by, referencing the first list. You then sort the new list and use it to figure out the order that the first list should be in. Here's a simple example that sorts a list of strings by length:

```
@sorted_by_length =
    map { $_ => [0] }           # Extract original list
    sort { $a=>[1] <=> $b=>[1] } # Sort by the transformed value
    map { [$_, length($_)] }   # Map to a list of element lengths
    @list;
```

Because each operator acts on the thing immediately to the right of it, it helps to read this from right to left (or bottom to top, the way it is written here).

The first thing that acts on the list is the `map` operator. It transforms the list into a hash in which the keys are the list elements and the values are the lengths of each element. This is where you put in your code that does the transformation by which you want to sort.

The next operator is the `sort` function, which sorts the list by the values.

Finally, the hash is transformed back into an array by extracting its keys. The array is now in the desired order.

Command-Line Processing

Perl is great at parsing the output of various programs. This is a task for which many people use tools such as `awk` and `sed`. Perl gives you a larger vocabulary for performing these tasks. The following example is very simple but illustrates how you might use Perl to chop up some output and do something with it. In this example, Perl is used to list only files that are larger than 10KB:

```
matthew@seymour:~$ ls -la | perl -nae 'print "%F[8] is %F[4]\n" if %F[4] > 10000;'
```

The `-n` switch indicates that the Perl code should run for each line of the output. The `-a` switch automatically splits the output into the `@F` array. The `-e` switch indicates that the Perl code is going to follow on the command line.

Related Ubuntu and Linux Commands

You use these commands and tools often when using Perl with Linux:

- ▶ `a2p`—A filter used to translate `awk` scripts into Perl
- ▶ `find2perl`—A utility used to create Perl code from command lines using the `find` command
- ▶ `perldoc`—A Perl utility used to read Perl documentation
- ▶ `s2p`—A filter used to translate `sed` scripts into Perl
- ▶ `vi`—The `vi` (actually `vim`) text editor

References

- ▶ ***Learning Perl, Third Edition*** by Randal L. Schwartz and Tom Phoenix—The standard entry text for learning Perl.
- ▶ ***Programming Perl***, 3rd edition, by Larry Wall, Tom Christiansen, and Jon Orwant—The standard advanced text for learning Perl.
- ▶ ***Mastering Regular Expressions*** by Jeffrey Friedl—Regular expressions are what make Perl so powerful.
- ▶ **www.perl.com**—This is the place to find all sorts of information about Perl, from its history and culture to helpful tips. This is also the place to download the Perl interpreter for your system.
- ▶ **<https://cpan.perl.org>**—CPAN is the place for you to find modules and programs in Perl. If you write something in Perl that you think is particularly useful, you can make it available to the Perl community here.
- ▶ **<https://perldoc.perl.org/index-faq.html>**—FAQ index of common Perl queries; this site offers a handy way to quickly search for answers about Perl.
- ▶ **<https://learn.perl.org>**—One of the best places to start learning Perl online.
- ▶ **<http://jobs.perl.org>**—If you master Perl, go to this site to look for a job.
- ▶ **www.pm.org**—The Perl Mongers are local Perl user groups. There might be one in your area.
- ▶ **www.perl.org**—This is the Perl advocacy site.

BONUS CHAPTER 43

Using Python

As PHP has come to dominate the world of web scripting, Python is increasingly dominating the domain of command-line scripting. Python's precise and clean syntax makes it one of the easiest languages to learn, and it allows programmers to code more quickly and spend less time maintaining their code. Whereas PHP is fundamentally similar to Java and Perl, Python is closer to C and Modula-3, and so it might look unfamiliar at first.

This chapter constitutes a quick-start tutorial to Python, designed to give you all the information you need to put together basic scripts and to point you toward resources that can take you further.

Two versions of Python are out in the wild right now. Python version 2.x reached end-of-life in 2019, although you are likely to see it in the wild for a while longer. Python 3.x is now the default version everywhere, including in Ubuntu, although Python 2.x will likely not yet be completely removed for Ubuntu 20.04; it will have to be installed intentionally alongside Python 3.x if it is needed.

Python 3.x is not backward compatible, and most programs written in or for 2.x need some work to run on 3.x, which is why the previous version is still available. However, most companies are working hard to update their code, such as LinkedIn, which announced in January 2020 that it “retired Python 2 and improved developer happiness” (<https://engineering.linkedin.com/blog/2020/how-we-retired-python-2-and-improved-developer-happiness>).

This chapter tries to note places where significant differences exist between 2.x and 3.x. If you are learning Python for the first time, start with a look at the 3.x series because it is the future.

IN THIS CHAPTER

- ▶ Python on Linux
- ▶ The Basics of Python
- ▶ Functions
- ▶ Object Orientation
- ▶ The Standard Library and the Python Package Index
- ▶ References

NOTE

This chapter provides a very elementary introduction to Python. For that reason, nearly everything we cover is identical in both 2.x and 3.x. We mention the differences earlier and in a couple of notes that follow so that readers who want to learn more will not be surprised later. Unless noted otherwise, you should find what we say in this chapter applies to either version.

Python on Linux

Most versions of Linux and UNIX, including macOS, come with Python preinstalled. This is partially for convenience because it is such a popular scripting language—and it saves having to install it later if the user wants to run a script—and partially because many vital or useful programs included in Linux distributions are written in Python. For example, the Ubuntu Software Center is a Python program.

The Python binary is installed in `/usr/bin/python` (or `/usr/bin/python3`); if you run that, you enter the Python interactive interpreter, where you can type commands and have them executed immediately. Although PHP also has an interactive mode (use `php -a` to activate it), it is neither as powerful nor as flexible as Python's.

As with Perl, PHP, and other scripting languages, you can also execute Python scripts by adding a shebang line (`#!`) to the start of your scripts that point to `/usr/bin/python` and then setting the file to be executable.

The third way to run Python scripts is through `mod_python`, which is an Apache module that embeds the Python interpreter into the HTTP server, allowing you to use Python to write web applications. You can install this from the Ubuntu repositories.

We use the interactive Python interpreter for this chapter because it provides immediate feedback on commands as you type them, so it is essential that you become comfortable using it. To get started, open a terminal and run the command `python`. You should see something like this, perhaps with different version numbers and dates:

```
matthew@seymour:~$ python
Python 3.6.4 (default, Dec27 2017, 13:02:49)
[GCC 7.2.0] on linux
Type "help", "copyright", "credits" or "license" for more information.
>>>
```

The `>>>` is where you type your input, and you can set and get a variable like this:

```
>>> python = 'great'
>>> python'great'
great
>>>
```

On the first line in this example, the variable `python` is set to the text `great`, and on the second line, you read back that value from the variable simply by typing the name

of the variable you want to read. The third line shows Python printing the variable; on the fourth line, you are back at the prompt to type more commands. Python remembers all the variables you use while in the interactive interpreter, which means you can set a variable to be the value of another variable.

When you are done, press Ctrl+D to exit. At this point, all your variables and commands are forgotten by the interpreter, which is why complex Python programs are always saved in scripts.

The Basics of Python

Python is a language wholly unlike most others, and yet it is so logical that most people can pick it up quickly. You have already seen how easily you can assign strings, but in Python, nearly everything is that easy—as long as you remember the syntax.

43

Numbers

The way Python handles numbers is more precise than in some other languages. It has all the normal operators—such as + for addition, - for subtraction, / for division, and * for multiplication—but it adds % for modulus (division remainder), ** for raise to the power, and // for floor division. It is also specific about which type of number is being used, as this example shows:

```
>>> a = 5
>>> b = 10
>>> a * b
50
>>> a / b
0
>>> b = 10.0
>>> a / b
0.5
>>> a // b
0.0
```

The first division returns 0 because both `a` and `b` are integers (whole numbers), so Python calculates the division as an integer, giving 0. By converting `b` to `10.0`, Python considers it to be a floating-point number, and so the division is now calculated as a floating-point value, giving 0.5. Even with `b` being a floating point, using `//`—floor division—rounds it down.

Using `**`, you can easily see how Python works with integers:

```
>>> 2 ** 30
1073741824
>>> 2 ** 31
2147483648L
```


As you saw in the previous example, you can assign a value to strings in Python with just an equal sign, like this:

```
>>> mystring = 'hello';
>>> myotherstring = "goodbye";
>>> mystring'hello'
>>> myotherstring;'goodbye'
>>> test = "Are you really Jayne Cobb?"
>>> test
"Are you really Jayne Cobb?"
```

The first example encapsulates the string in single quotation marks, and the second and third examples encapsulate the string in double quotation marks. However, printing the first and second strings shows them both in single quotation marks because Python does not distinguish between the two types. The third example is the exception; it uses double quotation marks because the string itself contains a single quotation mark. Here, Python prints the string with double quotation marks because it knows it contains the single quotation mark.

Because the characters in a string are stored in sequence, you can index into them by specifying the character you are interested in. As in most other languages, in Python these indexes are zero based, which means you need to ask for character 0 to get the first letter in a string. Here is an example:

```
>>> string = "This is a test string"
>>> string
'This is a test string'
>>> string[0]
'T'
>>> string [0], string[3], string [20]
('T', 's', 'g')
```

The last line shows how, with commas, you can ask for several indexes at the same time. You could print the entire first word using the following:

```
>>> string[0], string[1], string[2], string[3]
('T', 'h', 'I', 's')
```

However, for that purpose, you can use a different concept: slicing. A *slice* of a sequence draws a selection of indexes. For example, you can pull out the first word like this:

```
>>> string[0:4]
'This'
```

The syntax here means “take everything from position 0 (including 0) and end at position 4 (excluding it).” So, [0:4] copies the items at indexes 0, 1, 2, and 3. You can omit either side of the indexes, and it will copy either from the start or to the end:

```
>>> string [:4]
```

```
'This'
>>> string [5:]
'is a test string'
>>> string [11:]
'est string'
```

You can also omit both numbers, and it gives you the entire sequence:

```
>>> string[:] 'This is a test string'
```

Later you learn precisely why you would want to do this, but for now we look at a number of other string intrinsics that will make your life easier. For example, you can use the `+` and `*` operators to concatenate (join) and repeat strings, like this:

```
>>> mystring = "Python"
>>> mystring * 4
'PythonPythonPythonPython'
>>> mystring = mystring + " rocks! "
>>> mystring * 2
'Python rocks! Python rocks! '
```

In addition to working with operators, Python strings come with a selection of built-in methods. You can change the case of the letters with `capitalize()` (uppercases the first letter and lowercases the rest), `lower()` (lowercases them all), `title()` (uppercases the first letter in each word), and `upper()` (uppercases them all). You can also check whether strings match certain cases with `islower()`, `istitle()`, and `isupper()`; this also extends to `isalnum()` (which returns `true` if the string is letters and numbers only) and `isdigit()` (which returns `true` if the string is all numbers).

This example demonstrates some of these in action:

```
>>> string
'This is a test string'
>>> string.upper()
'THIS IS A TEST STRING'
>>> string.lower()
'this is a test string'
>>> string.isalnum()
False
>>> string = string.title()
>>> string
'This Is A Test String'
```

Why did `isalnum()` return `false`? Doesn't the string contain only alphanumeric characters? Well, no. There are spaces in there, which is what is causing the problem. More importantly, this example calls `upper()` and `lower()`, and they are not changing the contents of the string but just returning the new value. So, to change the string from `This`

is a test string to This Is A Test String, you have to assign it back to the string variable.

Another really useful and kind of cool thing you can do with strings is triple quoting. This way, you can easily create strings that include both double- and single-quoted strings, as well as strings that span more than one line, without having to escape a bunch of special characters. Here's an example:

```
>>>foo = """
... "foo"
... 'bar'
... baz
... blippy
... """
>>>foo
'\n"foo"\n\'bar\'\nbaz\nblippy\n'
>>>
```

Although this is intended as an introductory guide, the capability to use negative indexes with strings and slices is a pretty neat feature that deserves a mention. Here's an example:

```
>>>bar="news.google.com"
>>>bar[-1]
'm'
>>>bar[-4:]
'.com'
>>>bar[-10:-4]
'google'
>>>
```

One difference between Python 2.x and 3.x is the introduction of new string types and method calls. This doesn't affect what we share here but is something to pay attention to as you work with Python.

Lists

Python's built-in list data type is a sequence, like strings. However, lists are mutable, which means you can change them. A list is like an array in that it holds a selection of elements in a given order. You can cycle through them, index into them, and slice them:

```
>>> mylist = ["python", "perl", "php"]
>>> mylist
['python', 'perl', 'php']
>>> mylist + ["java"]
["python", 'perl', 'php', 'java']
>>> mylist * 2
['python', 'perl', 'php', 'python', 'perl', 'php']
>>> mylist[1]
```

```
'perl'
>>> mylist[1] = "c++"
>>> mylist[1]
'c++'
>>> mylist[1:3]
['c++', 'php']
```

The brackets notation is important: You cannot use parentheses `()` or braces `{}` for lists. Using `+` for lists is different from using `+` for numbers. Python detects that you are working with a list and appends one list to another. This is known as *operator overloading*, and it is part of what makes Python so flexible.

Lists can be *nested*, which means you can put a list inside a list. However, this is where mutability starts to matter, and so this might sound complicated. Recall that an immutable string sequence is a collection of characters that, once set, cannot be changed without creating a new string. Lists are mutable, as opposed to immutable, which means you can change a list without creating a new list. This becomes important because Python, by default, copies only a reference to a variable rather than the full variable. Consider this example:

```
>>> list1 = [1, 2, 3]
>>> list2 = [4, list1, 6]
>>> list1
[1, 2, 3]
>>> list2
[4, [1, 2, 3], 6]
```

This example shows a nested list. `list2` contains 4, then `list1`, then 6. When you print the value of `list2`, you can see that it also contains `list1`. Now, let's proceed on from that:

```
>>> list1[1] = "Flake"
>>> list2
[4, [1, 'Flake', 3], 6]
```

In the first line, you set the second element in `list1` (remember that sequences are zero based!) to be `Flake` rather than 2; then you print the contents of `list2`. As you can see, when `list1` changes, `list2` is updated, too. The reason for this is that `list2` stores a reference to `list1` as opposed to a copy of `list1`; they share the same value.

You can see that this works both ways by indexing twice into `list2`, like this:

```
>>> list2[1][1] = "Caramello"
>>> list1
[1, 'Caramello', 3]
```

The first line says, “Get the second element in `list2` (`list1`) and the second element of that list and set it to be ‘Caramello’.” Then `list1`'s value is printed, and you can see that

it has changed. This is the essence of mutability: You are changing a list without creating a new list. However, editing a string creates a new string, leaving the old one unaltered. Here is an example:

```
>>> mystring = "hello"
>>> list3 = [1, mystring, 3]
>>> list3
[1, 'hello', 3]
>>> mystring = "world"
>>> list3
[1, 'hello', 3]
```

Of course, this raises the question of how you copy without references when references are the default. The answer, for lists, is that you use the `[:]` slice, covered earlier in this chapter. This slices from the first element to the last, inclusive, essentially copying it without references. Here is how it looks:

```
>>> list4 = ["a", "b", "c"]
>>> list5 = list4[:]
>>> list4 = list4 + ["d"]
>>> list5
['a', 'b', 'c']
>>> list4
['a', 'b', 'c', 'd']
```

Lists have their own collections of built-in methods, such as `sort()`, `append()`, and `pop()`. The latter two add and remove single elements from the end of the list, and `pop()` also returns the removed element. Here is an example:

```
>>> list5 = ["nick", "paul", "Julian", "graham"]
>>> list5.sort()
>>> list5
['graham', 'julian', 'nick', 'paul']
>>> list5.pop() 'paul'
>>> list5
['graham', 'julian', 'nick']

>>> list5.append("Rebecca")
```

In addition, one interesting method of strings returns a list: `split()`. This takes a character to split by and then gives you a list in which each element is a chunk from the string. Here is an example:

```
>>> string = "This is a test string";
>>> string.split(" ")
['This', 'is', 'a', 'test', 'string']
```

Lists are used extensively in Python, although this is slowly changing as the language matures. The way lists are compared and sorted has changed in 3.x, so be sure to check the latest documentation when you attempt to perform these tasks.

Dictionaries

Unlike a list, a dictionary is a collection with no fixed order. Instead, it has a *key* (the name of the element) and a *value* (the content of the element), and Python places a dictionary wherever it needs to for maximum performance. When defining dictionaries, you need to use braces (`{}`) and colons (`:`). You start with an opening brace and then give each element a key and a value, separated by a colon, like this:

```
>>> mydict = { "perl" : "a language", "php" : "another language" }
>>> mydict
{'php': 'another language', 'perl': 'a language'}
```

This example has two elements, with keys `perl` and `php`. However, when the dictionary is printed, you find that `php` comes before `perl`; Python hasn't respected the order in which you entered them. You can index into a dictionary by using the normal code:

```
>>> mydict["perl"] 'a language'
```

However, because a dictionary has no fixed sequence, you cannot take a slice or an index by position.

Like lists, dictionaries are mutable and can also be nested; however, unlike with lists, you cannot merge two dictionaries by using `+`. This is because dictionary elements are located using the key. Therefore, having two elements with the same key would cause a clash. Instead, you should use the `update()` method, which merges two arrays by overwriting clashing keys.

You can also use the `keys()` method to return a list of all the keys in a dictionary.

Subtle differences exist between how dictionaries are used in 3.x compared to in 2.x, such as the need to make list calls to produce all values at once so they may be printed. Again, be sure to check the latest documentation as you move ahead because there are several of these changes in functionality, with some tools now behaving differently or even disappearing and other new dictionary tools being added. Some of those features have been backported; for example, Python 2.7 received support for ordered dictionaries, backported from Python 3.1 (see <https://docs.python.org/dev/whatsnew/2.7.html#pep-0372>).

Conditionals and Looping

So far, we have just been looking at data types, which should show you how powerful Python's data types are. However, you cannot write complex programs without conditional statements and loops.

Python has most of the standard conditional checks, such as `>` (greater than), `<=` (less than or equal to), and `==` (equal), but it also adds some new ones, such as `in`. For example, you can use `in` to check whether a string or a list contains a given character or element:

```
>>> mystring = "J Random Hacker"
>>> "r" in mystring
True
>>> "Hacker" in mystring
True
>>> "hacker" in mystring
False
```

This example demonstrates how strings are case sensitive. You can use the equal operator for lists, too:

```
>>> mylist = ["soldier", "sailor", "tinker", "spy"]
>>> "tailor" in mylist
False
```

Other comparisons on these complex data types are done item by item:

```
>>> list1 = ["alpha", "beta", "gamma"]
>>> list2 = ["alpha", "beta", "delta"]
>>> list1 > list2
True
```

In this case, `list1`'s first element (`alpha`) is compared against `list2`'s first element (`alpha`) and, because they are equal, the next element is checked. That is equal also, so the third element is checked, which is different. The `g` in `gamma` comes after the `d` in `delta` in the alphabet, so `gamma` is considered greater than `delta`, and `list1` is considered greater than `list2`.

Loops come in two types, and both are equally flexible. For example, the `for` loop can iterate through letters in a string or elements in a list:

```
>>> string = "Hello, Python!"
>>> for s in string: print s,
...
H e l l o ,   P y t h o n !
```

The `for` loop takes each letter in a string and assigns it to `s`. This is then printed to the screen using the `print` command, but note the comma at the end; it tells Python not to insert a line break after each letter. The ellipsis (`...`) is there because Python allows you to enter more code in the loop; you need to press Enter again here to have the loop execute.

You can use the same construct for lists:

```
>>> mylist = ["andi", "rasmus", "zeev"]
>>> for p in mylist: print p
```



```
...
andi
rasmus
zeev
```

Without the comma after the `print` statement, each item is printed on its own line.

The other loop type is the `while` loop, and it looks similar:

```
>> while 1: print "This will loop forever!"
...
This will loop forever!
This will loop forever!
This will loop forever!
This will loop forever!
This will loop forever!
(etc)
Traceback (most recent call last):
  File "<stdin>", line 1, in ?
KeyboardInterrupt
>>>
```

That is an infinite loop (it carries on printing that text forever), so you need to press `Ctrl+C` to interrupt it and regain control.

If you want to use multiline loops, you need to get ready to use your `Tab` key: Python handles loop blocks by recording the level of indent used. Some people find this odious; others admire it for forcing clean coding on users. Most of us, though, just get on with programming.

Here is an example:

```
>>> i = 0
>>> while i < 3:
...     j = 0
...     while j < 3:
...         print "Pos: " + str(i) + "," + str(j) + " "
...         j += 1
...     i += 1
...
Pos: (0,0)
Pos: (0,1)
Pos: (0,2)
Pos: (1,0)
Pos: (1,1)
Pos: (1,2)
Pos: (2,0)
Pos: (2,1)
Pos: (2,2)
```

You can control loops by using the `break` and `continue` keywords. `break` exits the loop and continues processing immediately afterward, and `continue` jumps to the next loop iteration.

Again, many subtle changes have occurred from 2.x to 3.x. Most conditionals and looping work the same, but because we mention it in this section, we should note that `print` is no longer a statement in 3.x and is now a function call. That provides a nice segue to our discussion of functions.

Functions

Some languages—such as PHP—read and process an entire file before executing it, which means you can call a function before it is defined because the compiler reads the definition of the function before it tries to call it. Python is different: If the function definition has not been reached by the time you try to call it, you get an error. The reason behind this behavior is that Python actually creates an object for your function, and that in turn means two things. First, you can define the same function several times in a script and have the script pick the right one at runtime. Second, you can assign the function to another name just by using `=`.

A function definition starts with `def`, then the function name, then parentheses and a list of parameters, and then a colon. The contents of a function need to be indented at least one level beyond the definition. So, using function assignment and dynamic declaration, you can write a script that prints the right greeting in a roundabout manner:

```
>>> def hello_english(Name):
...     print "Hello, " + Name + "!"
...
>>> def hello_hungarian(Name):
...     print "Szia, " + Name + "!"
...
>>> hello = hello_hungarian
>>> hello("Paul")
Szia, Paul!
>>> hello = hello_english
>>> hello("Paul")
```

Notice that function definitions include no type information. Functions are typeless, as mentioned previously. The upside of this is that you can write one function to do several things:

```
>>> def concat(First, Second):
...     return First + Second
...
>>> concat(["python"], ["perl"])
['python', 'perl']
>>> concat("Hello, ", "world!")'Hello, world!'
```

This example demonstrates how the `return` statement sends a value back to the caller and also how a function can do one thing with lists and another thing with strings. The magic here is being accomplished by the objects. You write a function that tells two objects to add themselves together, and the objects intrinsically know how to do that. If they do not—if, perhaps, the user passes in a string and an integer—Python catches the error for you. However, it is this hands-off, “let the objects sort themselves out” attitude that makes functions so flexible. The `concat()` function could conceivably concatenate strings, lists, or *zonks*—a data type someone created herself that allows adding. The point is that Python does not limit what your function can do. As clichéd as it might sound, the only limit is your imagination.

Object Orientation

After having read this far, you should not be surprised to hear that Python’s object orientation is flexible and likely to surprise you if you have been using C-like languages for some time.

The best way to learn Python *object-oriented programming (OOP)* is to just do it. Here is a basic script that defines a class, creates an object of that class, and calls a function:

```
class Dog(object):
    def bark(self):
        print "Woof!"

fluffy = Dog()
fluffy.bark()
```

Defining a class starts, predictably, with the `class` keyword followed by the name of the class you are defining and a colon. The contents of that class need to be indented one level so that Python knows where it stops. Note that the object inside parentheses is there for object inheritance, which is discussed later. For now, the least you need to know is that if your new class is not based on an existing class, you should put `object` inside parentheses, as shown in the previous code.

Functions inside classes work in much the same way as normal functions do (although they are usually called *methods*); the main difference is that they should all take at least one parameter, usually called `self`. This parameter is filled with the name of the object the function was called on, and you need to use it explicitly.

Creating an instance of a class is done by assignment. You do not need any new keyword, as in some other languages; providing the parentheses makes the creation known. Calling a function of that object is done using a period and the name of the class to call, with any parameters being passed inside parentheses.

Class and Object Variables

Each object has its own set of functions and variables, and you can manipulate those variables independently of objects of the same type. In addition, a class variable may be set to a default value for all classes, in which case it can be manipulated globally.

This script demonstrates two objects of the `Dog` class being created, each with its own name:

```
class Dog(object):
    name = "Lassie"
    def bark(self):
        print self.name + " says 'Woof!'"
    def set_name(self, name):
        self.name = name

fluffy = Dog()
fluffy.bark()
poppy = Dog()
poppy.set_name("Poppy")
poppy.bark()
```

This outputs the following:

```
Lassie says 'Woof!'
Poppy says 'Woof!'
```

Here, each dog starts with the name `Lassie`, but it gets customized. Keep in mind that, by default, Python assigns by reference, meaning each object has a reference to the class's name variable, and as you assign that with the `set_name()` method, that reference is lost. This means that any references you do not change can be manipulated globally. Thus, if you change a class's variable, it also changes in all instances of that class that have not set their own value for that variable. Consider this example:

```
class Dog(object):
    name = "Lassie"
    color = "brown"

fluffy = Dog()
poppy = Dog()
print fluffy.color
Dog.color = "black"
print poppy.color
fluffy.color = "yellow"
print fluffy.color
print poppy.color
```

So, the default color of dogs is `brown`—both the `fluffy` and `poppy` `Dog` objects start off as `brown`. Then, using `Dog.color`, you set the default color to be `black`, and because neither of the two objects has set its own color value, they are updated to be `black`. The third-to-last line uses `poppy.color` to set a custom color value for the `poppy` object: `poppy` becomes `yellow`, whereas `fluffy` and the `Dog` class in general remain `black`.

Constructors and Destructors

To help you automate the creation and deletion of objects, you can easily override two default methods: `__init__` and `__del__`. These are the methods that Python calls when a class is being instantiated and freed, known as the *constructor* and *destructor*, respectively.

Having a custom constructor is great for when you need to accept a set of parameters for each object being created. For example, you might want each dog to have its own name on creation, and you could implement that with this code:

```
class Dog(object):
    def __init__(self, name):
        self.name = name
fluffy = Dog("Fluffy")
print fluffy.name
```

If you do not provide a name parameter when creating the `Dog` object, Python reports an error and stops. You can, of course, ask for as many constructor parameters as you want, although it is usually better to ask only for the ones you need and have other functions to fill in the rest.

On the other side of things is the destructor method, which enables you to have more control over what happens when an object is destroyed. Using a constructor and destructor, this example shows the life cycle of an object by printing messages when it is created and deleted:

```
class dog(object):
    def __init__(self, name):
        self.name = name
        print self.name + " is alive!"
    def __del__(self):
        print self.name + " is no more!"
fluffy = Dog("Fluffy")
```

The destructor is here to give you the chance to free up resources allocated to the object or perhaps log something to a file. Note that although it is possible to override the `__del__` method, doing so is very dangerous because it is not guaranteed that `__del__` methods are called for objects that still exist when the interpreter exits. The official Python documentation explains this well at https://docs.python.org/reference/datamodel.html#object.__del__.

Class Inheritance

Python allows you to reuse your code by inheriting one class from one or more others. For example, lions, tigers, and bears are all mammals and so share a number of similar properties. In that scenario, you do not want to have to copy and paste functions between them; it is smarter (and easier) to have a mammal class that defines all the shared functionality and then inherit each animal from that.

Consider the following code:

```
class Car(object):
    color = "black"
    speed = 0
    def accelerate_to(self, speed):
        self.speed = speed
    def set_color(self, color):
```

```
        self.color = color
mycar = Car()
print mycar.color
```

This example creates a `Car` class with a default color and provides a `set_color()` function so that people can change their own colors. Now, what do you drive to work? Is it a car? Sure it is, but chances are it is a Ford, or a Dodge, or a Jeep, or some other make; you do not get cars without a make. On the other hand, you do not want to have to define a class `Ford` and give it the methods `accelerate_to()`, `set_color()`, and however many other methods a basic car has and then do the same thing for Ferrari, Nissan, and so on.

The solution is to use inheritance: Define a `car` class that contains all the shared functions and variables of all the makes and then inherit from that. In Python, you do this by putting the name of the class from which to inherit inside parentheses in the `class` declaration, like this:

```
class Car(object):
    color = "black"
    speed = 0
    def accelerate_to(self, speed):
        self.speed = speed
    def set_color(self, color):
        self.color = color
class Ford(Car): pass
class Nissan(Car): pass
mycar = Ford()
print mycar.color
```

The `pass` directive is an empty statement; it means the class contains nothing new. However, because the `Ford` and `Nissan` classes inherit from the `car` class, they get `color`, `speed`, `accelerate_to()`, and `set_color()` provided by their parent class. (Note that you do not need objects after the class names for `Ford` and `Nissan` because they are inherited from an existing class: `car`.)

By default, Python gives you all the methods the parent class has, but you can override that by providing new implementations for the classes that need them. Here is an example:

```
class Modelt(car):
    def set_color(self, color):
        print "Sorry, Model Ts only come in black!"

myford = Ford()
Ford.set_color("green")
mycar = Modelt()
mycar.set_color("green")
```

The first car is created as a `Ford`, so `set_color()` works fine because it uses the method from the `car` class. However, the second car is created as a `Modelt`, which has its own `set_color()` method, so the call fails.

This suggests an interesting scenario: What do you do if you have overridden a method and yet want to call the parent's method also? If, for example, changing the color of a `Model T` were allowed but just cost extra, you would want to print a message saying "You owe \$50 more" but then change the color. To do this, you need to use the `class` object from which the current class is inherited (`Car` in this case). Here's an example:

```
class Modelt(Car):
    def set_color(self, color):
        print "You owe $50 more"
        Car.set_color(self, color)

mycar = Modelt()
mycar.set_color("green")
print mycar.color
```

This prints the message and then changes the color of the car.

The Standard Library and the Python Package Index

A default Python install includes many modules (blocks of code) that enable you to interact with the operating system, open and manipulate files, parse command-line options, perform data hashing and encryption, and much more. This is one of the reasons most commonly cited when people are asked why they like Python so much: It comes stocked to the gills with functionality you can take advantage of immediately. In fact, the number of modules included in the Standard Library is so high that entire books have been written about them.

For unofficial scripts and add-ons for Python, the recommended starting place is called the *Python Package Index* (*PyPI*), at <https://pypi.python.org/pypi>. There you can find more than 10,000 public scripts and code examples for everything from mathematics to games.

References

- ▶ **www.python.org**—The Python website is packed with information and updated regularly. This should be your first stop, if only to read the latest Python news.
- ▶ **www.jython.org**—Python also has an excellent Java-based interpreter to allow it to run on other platforms.
- ▶ **www.ironpython.com**—If you prefer Microsoft .NET, check out this site.
- ▶ **www.montypython.com**—Guido van Rossum borrowed the name for his language from *Monty Python's Flying Circus*, and as a result many Python code examples use oblique *Monty Python* references (spam and eggs are quite common, for example). A visit to the official *Monty Python* site to hone your Python knowledge is highly recommended.
- ▶ **www.zope.org**—This is the home page of the *Zope content management system* (CMS). One of the most popular CMSs around, Zope is written entirely in Python.

BONUS CHAPTER 44

Using PHP

This chapter introduces you to the world of PHP programming from the point of view of using it as a web scripting language and as a command-line tool. PHP originally stood for *Personal Home Page* because it was a collection of Perl scripts designed to ease the creation of guest books, message boards, and other interactive scripts commonly found on home pages. However, since those early days, it has received major updates and revisions.

Part of the success of PHP has been its powerful integration with databases; its earliest uses nearly always took advantage of a database back end.

NOTE

Many packages for PHP are available from the Ubuntu repositories. The basic package you want to install is just called `php`, and installing it brings in other packages as dependencies.

Introduction to PHP

In terms of the way it looks, PHP is a cross between Java and Perl, a successful merger of the best aspects of both into one language. The Java parts include a powerful object-orientation system, the capability to throw program exceptions, and the general style of writing that both languages borrowed from C. Borrowed from Perl is the “it should just work” mentality where ease of use is favored over strictness. As a result, you will find a lot of “there is more than one way to do it” in PHP. This also means that it is possible to accomplish tasks in ways that are less than ideal or without consideration for good security. Many criticize PHP for this, but for simple tasks or if written carefully, it can be a pretty good language and is easy to understand and use, especially for quick website creation.

IN THIS CHAPTER

- ▶ Introduction to PHP
- ▶ Basic Functions
- ▶ Handling HTML Forms
- ▶ Databases
- ▶ References

Entering and Exiting PHP Mode

Unlike with PHP's predecessors, you embed your PHP code inside your HTML as opposed to the other way around. Before PHP, many websites had standard HTML pages for most of their content, linking to Perl CGI pages to do back-end processing when needed. With PHP, all your pages are capable of processing and containing HTML. This is a huge factor in PHP's popularity.

Each PHP file is processed by PHP, which looks for code to execute. PHP considers all the text it finds to be HTML until it finds one of four things:

- ▶ `<?php`
- ▶ `<?`
- ▶ `<%`
- ▶ `<script language="php">`

The first option is the preferred method of entering PHP mode because it is guaranteed to work.

When in PHP mode, you can exit it by using `?>` (for `<?php` and `<?`); `%>` (for `<%`); or `</script>` (for `<script language="php">`). This code example demonstrates entering and exiting PHP mode:

In HTML mode

```
<?php
    echo "In PHP mode";
?>
In HTML mode
In <?php echo "PHP"; ?> mode
```

Variables

Every variable in PHP starts with a dollar sign (`$`). Unlike many other languages, PHP does not have different types of variable for integers, floating-point numbers, arrays, or Booleans. They all start with a `$`, and all are interchangeable. As a result, PHP is a weakly typed language, which means you do not declare a variable as containing a specific type of data; you just use it however you want to.

Save the code in Listing 44.1 into a new file called `ubuntu1.php`.

LISTING 44.1 Testing Types in PHP

```
<?php
    $i = 10;
    $j = "10";
    $k = "Hello, world";
    echo $i + $j;
    echo $i + $k;
?>
```

To run this script, bring up a console and browse to where you saved it. Then type this command:

```
matthew@seymour:~$ php ubuntu1.php
```

If PHP is installed correctly, you should see the output `2010`, which is really two things. The `20` is the result of `10 + 10` (`$i` plus `$j`), and the `10` is the result of adding `10` to the text string `Hello, world`. Neither of those operations are really straightforward. Whereas `$i` is set to the number `10`, `$j` is actually set to be the text value `"10"`, which is not the same thing. Adding `10` to `10` gives `20`, as you would imagine, but adding `10` to `"10"` (the string) forces PHP to convert `$j` to an integer on-the-fly before adding it.

Running `$i + $k` adds another string to a number, but this time the string is `Hello, world` and not just a number inside a string. PHP still tries to convert it, though, and converting any non-numerical string into a number converts it to `0`. So, the second `echo` statement ends up saying `$i + 0`.

As you should have guessed by now, calling `echo` outputs values to the screen. Right now, that prints directly to your console, but internally PHP has a complex output mechanism that enables you to print to a console, send text through Apache to a web browser, send data over a network, and more.

Now that you have seen how PHP handles variables of different types, it is important that you understand the various of types available to you, as shown in Table 44.1.

Table 44.1 PHP Variable Types

Type	Stores
integer	Whole numbers; for example, <code>1</code> , <code>9</code> , or <code>324809873</code>
float	Fractional numbers; for example, <code>1.1</code> , <code>9.09</code> , or <code>3.141592654</code>
string	Characters; for example, <code>"a"</code> , <code>"sfdgh"</code> , or <code>"Ubuntu Unleashed"</code>
boolean	<code>true</code> or <code>false</code>
array	Several variables of any type

The first four variables in Table 44.1 can be thought of as simple variables and the last two as complex variables. Arrays are simply collections of variables. You might have an array of numbers (for example, the ages of all the children in a class); an array of strings (for example, the names of all Wimbledon tennis champions); or even an array of arrays, known as a *multidimensional array*. Arrays are covered in more depth in the next section because they are unique in the way they are defined.

An object is used to define and manipulate a set of variables that belong to a unique entity. Each object has its own personal set of variables, as well as functions that operate on those variables. Objects are commonly used to model real-world things. You might define an object that represents a TV, with variables such as `$CurrentChannel` (probably an integer), `$SupportsHiDef` (a Boolean), and so on.

Of all the complex variables, the easiest to grasp are resources. PHP has many extensions available to it that allow you to connect to databases, manipulate graphics, or even make

calls to Java programs. Because they are all external systems, they need to have types of data unique to them that PHP cannot represent using any of the six other data types. So, PHP stores their custom data types in resources—data types that are meaningless to PHP but can be used by the external libraries that created them.

Arrays

Arrays are one of our favorite parts of PHP because the syntax is smart and easy to read and yet manages to be as powerful as you could want. You need to know four pieces of jargon to understand arrays:

- ▶ An array is made up of many *elements*.
- ▶ Each element has a *key* that defines its place in the array. An array can have only one element with a given key.
- ▶ Each element also has a *value*, which is the data associated with the key.
- ▶ Each array has a *cursor*, which points to the current key.

The first three are used regularly; the last one less often. The array cursor is covered later in this chapter, in the section “Basic Functions,” but we look at the other three now. With PHP, your keys can be almost anything: integers, strings, objects, or other arrays. You can even mix and match the keys so that one key is an array, another is a string, and so on. The one exception to all this is floating-point numbers; you cannot use floating-point numbers as keys in your arrays.

There are two ways of adding values to an array: with the `[]` operator, which is unique to arrays; and with the `array()` pseudo-function. You should use `[]` when you want to add items to an existing array and use `array()` to create a new array.

To sum all this up in code, Listing 44.2 shows a script that creates an array without specifying keys, adds various items to it both without keys and with keys of varying types, does a bit of printing, and then clears the array.

LISTING 44.2 Manipulating Arrays

```
<?php
    $myarr = array(1, 2, 3, 4);

    $myarr[4] = "Hello";
    $myarr[] = "World!";
    $myarr["elephant"] = "Wombat";
    $myarr["foo"] = array(5, 6, 7, 8);

    echo $myarr[2];
    echo $myarr["elephant"];
    echo $myarr["foo"][1];

    $myarr = array();

?>
```

The initial array is created with four elements, to which we assign the values 1, 2, 3, and 4. Because no keys are specified, PHP automatically assigns keys for us, starting at 0 and counting upward—giving keys 0, 1, 2, and 3. Then we add a new element with the `[]` operator, specifying 4 as the key and "Hello" as the value. Next, `[]` is used again to add an element with the value "World!" and no key and then again to add an element with the key "elephant" and the value "wombat". The line after that demonstrates using a string key with an array value—an array inside an array (a multidimensional array).

The next three lines demonstrate reading back from an array, first using a numerical key, then using a string key, and then using a string key and a numerical key. Remember that the "foo" element is an array in itself, so that third reading line retrieves the array and then prints the second element (arrays start at 0, remember). The last line blanks the array by simply using `array()` with no parameters, which creates an array with elements and assigns it to `$myarr`.

The following is an alternative way of using `array()` that allows you to specify keys along with their values:

```
$myarr = array("key1" => "value1", "key2" => "value2",  
7 => "foo", 15 => "bar");
```

Which method you choose really depends on whether you want specific keys or want PHP to pick them for you.

Constants

Constants are frequently used in functions that require specific values to be passed in. For example, a popular function is `extract()`, which takes all the values in an array and places them into variables in their own right. You can choose to change the name of the variables as they are extracted by using the second parameter; send it 0, and it overwrites variables with the same names as those being extracted; send it 1, and it skips variables with the same names; send it 5, and it prefixes variables only if they exist already; and so on. Of course, no one wants to have to remember a lot of numbers for each function, so you can instead use `EXTR_OVERWRITE` for 0, `EXTR_SKIP` for 1, `EXTR_PREFIX_IF_EXISTS` for 5, and so on, which is much easier.

You can create constants of your own by using the `define()` function. Unlike variables, constants do not start with a dollar sign. Code to define a constant looks like this:

```
<?php  
    define("NUM_SQUIRRELS", 10);  
    define("PLAYER_NAME", "Jim");  
    define("NUM_SQUIRRELS_2", NUM_SQUIRRELS);  
    echo NUM_SQUIRRELS_2;  
?>
```

This script demonstrates how you can set constants to numbers, strings, or even the values of other constants (although this last option doesn't really get used much).

References

You can use the equal sign (=) to copy the value from one variable to another so that each one has a copy of the value. Another option here is to use references, which is where a variable does not have a value of its own but instead points to another variable. This enables you to share values and have variables mutually update themselves.

To copy by reference, use the & symbol, as follows:

```
<?php
    $a = 10;
    $b = &$a;
    echo $a . "\n";
    echo $b . "\n";

    $a = 20;
    echo $a . "\n";
    echo $b . "\n";

    $b = 30;
    echo $a . "\n";
    echo $b . "\n";
?>
```

If you run this script, you will see that updating `$a` also updates `$b` and also that updating `$b` updates `$a`.

Comments

Adding short comments to your code is recommended and usually a requirement in larger software houses. In PHP, you have three options for commenting style: `//`, `/* */`, and `#`. The first option (two slashes) instructs PHP to ignore everything until the end of the line. The second (a slash and an asterisk) instructs PHP to ignore everything until it reaches `*/`. The last (a hash symbol) works like `//` and is included because it is common among shell scripting languages.

This code example demonstrates the difference between `//` and `/* */`:

```
<?php
    echo "This is printed!";
    // echo "This is not printed";
    echo "This is printed!";
    /* echo "This is not printed";
    echo "This is not printed either"; */
?>
```

Using `//` is generally preferred because it is a known quantity. It is easy to introduce coding errors with `/* */` by losing track of where a comment starts and ends.

NOTE

Contrary to popular belief, having comments in your PHP script has almost no effect on the speed at which the script executes. What little speed difference exists is wholly removed if you use a code cache.

Escape Sequences

Some characters cannot be typed, and yet you will almost certainly want to use some of them from time to time. For example, you might want to use an ASCII character for a new line, but you cannot type it. Instead, you need to use an escape sequence: `\n`. Similarly, you can print a carriage return character with `\r`. It is important to know both of these because, on the Windows platform, you need to use `\r\n` to get a new line. If you do not plan to run your scripts anywhere other than your local Ubuntu or other Linux environment, you need not worry about this.

Going back to the first script you wrote, recall that it printed `2010` because you added `10 + 10` and then `10 + 0`. You can rewrite that using escape sequences, like this:

```
<?php
    $i = 10;
    $j = "10";
    $k = "Hello, world";
    echo $i + $j;
    echo "\n";
    echo $i + $k;
    echo "\n";
?>
```

This time PHP prints a new line after each of the numbers, making it obvious that the output is `20` and `10` rather than `2010`. Note that the escape sequences must be used in double quotation marks because they will not work in single quotation marks.

Three common escape sequences are `\\`, which means “ignore the backslash”; `\"`, which means “ignore the double quote”; and `\'`, which means “ignore the single quote.” This is important when strings include quotation marks inside them. If we had a string such as `"Are you really Conan O'Brien?"`, which has a single quotation mark in it, this code would not work:

```
<?php
    echo 'Are you really Conan O'Brien?';
?>
```

PHP would see the opening quotation mark, read all the way up to the `O` in `O'Brien`, and then see the quotation mark following the `O` as being the end of the string. The `Brien?` part would appear to be a fragment of text and would cause an error. You have two options here: You can either surround the string in double quotation marks or escape the single quotation mark with `\'`. The escaping route looks like this:

```
echo "Are you really Conan O\'Brien?";
```

Although escape sequences are a clean solution for small text strings, be careful not to overuse them. HTML is particularly full of quotation marks, and escaping them can get messy, as you can see in this example:

```
$mystring = "<img src=\"foo.png\" alt=\"My picture\"
width=\"100\" height=\"200\" />";
```

In such a situation, you are better off using single quotation marks to surround the text simply because it is a great deal easier on the eye.

Variable Substitution

PHP allows you to define strings using three methods: single quotation marks, double quotation marks, or heredoc notation. Heredoc is not discussed in this chapter because it is fairly rare compared to the other two methods, but single quotation marks and double quotation marks work identically, with one minor exception: variable substitution.

Consider the following code:

```
<?php
    $age = 25
    echo "You are ";
    echo $age;
?>
```

This is a particularly clumsy way to print a variable as part of a string. Fortunately, if you put a variable inside a string, PHP performs *variable substitution*, replacing the variable with its value. That means you can rewrite the code like this:

```
<?php
    $age = 25
    echo "You are $age";
?>
```

The output is the same. The difference between single quotation marks and double quotation marks is that single-quoted strings do not have their variables substituted. Here's an example:

```
<?php
    $age = 25
    echo "You are $age";
    echo 'You are $age';
?>
```

The first `echo` prints `You are 25`, and the second one prints `You are $age`.

Operators

Now that you have data values to work with, you need some operators to use, too. You have already used `+` to add variables together, but many other operators in PHP handle

arithmetic, comparison, assignment, and other operators. *Operator* is just a fancy word for something that performs an operation, such as addition or subtraction. However, *operand* might be new to you. Consider this operation:

```
$a = $b + c;
```

In this operation, = and + are operators, and \$a, \$b, and \$c are operands. Along with +, you also already know - (subtract), * (multiply), and / (divide), but Table 44.2 provides some more.

Table 44.2 PHP Operators

Operator	What It Does
=	Assigns the right operand to the left operand.
==	Returns <code>true</code> if the left operand is equal to the right operand.
!=	Returns <code>true</code> if the left operand is not equal to the right operand.
===	Returns <code>true</code> if the left operand is identical to the right operand. This is not the same as <code>==</code> .
!==	Returns <code>true</code> if the left operand is not identical to the right operand. This is not the same as <code>!=</code> .
<	Returns <code>true</code> if the left operand is smaller than the right operand.
>	Returns <code>true</code> if the left operand is greater than the right operand.
<=	Returns <code>true</code> if the left operand is equal to or smaller than the right operand.
&&	Returns <code>true</code> if both the left operand and the right operand are true.
	Returns <code>true</code> if either the left operand or the right operand is true.
++	Increments the operand by 1.
--	Decrements the operand by 1.
+=	Increments the left operand by the right operand.
-=	Decrements the left operand by the right operand.
.	Concatenates the left operand and the right operand (joins them together).
%	Divides the left operand by the right operand and returns the remainder.
	Performs a bitwise OR operation. It returns a number with bits that are set in either the left operand or the right operand.

At least 10 other operators are not listed in the table because you're unlikely to use them. Even some of the ones in the table are used infrequently (bitwise AND, for example). Having said that, the bitwise OR operator (|) is used regularly because it allows you to combine values.

Here is a code example that demonstrates some of the operators:

```
<?php
    $i = 100;
    $i++; // $i is now 101
    $i--; // $i is now 100 again
```



```

$i += 10; // $i is 110
$i = $i / 2; // $i is 55
$j = $i; // both $j and $i are 55
$i = $j % 11; // $i is 0
?>

```

The last line uses modulus, which takes some people a little bit of effort to understand. The result of `$i % 11` is 0 because `$i` is set to 55, and modulus works by dividing the left operand (55) by the right operand (11) and returning the remainder. 55 divides by 11 exactly 5 times, and so has no remainder, or 0.

The concatenation operator, a period (`.`), sounds scarier than it is: It just joins strings together. Here is an example:

```

<?php
echo "Hello, " . "world!";
echo "Hello, world!" . "\n";
?>

```

There are two special operators in PHP that are not covered here and yet are used frequently. Before we look at them, though, it is important that you see how the comparison operators (such as `<`, `<=`, and `!=`) are used inside conditional statements.

Conditional Statements

In a *conditional statement*, you instruct PHP to take different actions depending on the outcome of a test. For example, you might want PHP to check whether a variable is greater than 10 and, if so, print a message. This is all done with the `if` statement, which looks like this:

```

if (your condition) {
    // action to take if condition is true
} else {
    // optional action to take otherwise
}

```

The *your condition* part can be filled with any number of conditions you want PHP to evaluate, and this is where the comparison operators come into their own. Consider this example:

```

if ($i > 10) {
    echo "11 or higher";
} else {
    echo "10 or lower";
}

```

PHP looks at the condition and compares `$i` to `10`. If it is greater than `10`, it replaces the whole operation with `1`; otherwise, it replaces it with `0`. So, if `$i` is `20`, the result looks like this:

```
if (1) {
    echo "11 or higher";
} else {
    echo "10 or lower";
}
```

In conditional statements, any number other than `0` is considered to be equivalent to the Boolean value `true`; so `1` always evaluates to `true`. There is a similar case for strings: If your string has any characters in it, it evaluates to `true`, with empty strings evaluating to `false`. This is important because you can then use that `1` in another condition through `&&` or `||` operators. For example, if you want to check whether `$i` is greater than `10` but less than `40`, you could write this:

```
if ($i > 10 && $i < 40) {
    echo "11 or higher";
} else {
    echo "10 or lower";
}
```

If you presume that `$i` is set to `50`, the first condition (`$i > 10`) is replaced with `1`, and the second condition (`$i < 40`) is replaced with `0`. Those two numbers are then used by the `&&` operator, which requires both the left and right operands to be `true`. Whereas `1` is equivalent to `true`, `0` is not, so the `&&` operand is replaced with `0`, and the condition fails.

`=`, `==`, `===`, and similar operators are easily confused and often the source of programming errors. The first, a single equal sign, assigns the value of the right operand to the left operand. However, all too often you see code like this:

```
if ($i = 10) {
    echo "The variable is equal to 10!";
} else {
    echo "The variable is not equal to 10";
}
```

This code is incorrect. Rather than checking whether `$i` is equal to `10`, it assigns `10` to `$i` and returns `true`. What is needed is `==`, which compares two values for equality. In PHP, this is extended so that there is also `===` (three equal signs), which checks whether two values are identical, more than just equal.

The difference is slight but important: If you have a variable with the string value `"10"` and compare it against the number value of `10`, they are equal. Thus, PHP converts the type and checks the numbers. However, they are not identical. To be considered identical, the two variables must be equal (that is, have the same value) and be of the same data type (that is, both strings, both integers, and so on).

NOTE

Putting function calls in conditional statements rather than direct comparisons is common practice. Here is an example:

```
if (do_something()) {
```

If the `do_something()` function returns `true` (or something equivalent to `true`, such as a nonzero number), the conditional statement evaluates to `true`.

Special Operators

The ternary operator and the execution operator work differently from the operators you have seen so far. The ternary operator is rarely used in PHP, thankfully, because it is really just a condensed conditional statement. Presumably it arose through someone needing to make code occupy as little space as possible because it certainly does not make PHP code any easier to read.

The ternary operator works like this:

```
$age_description = ($age < 18) ? "child" : "adult";
```

Without explanation, this code is essentially meaningless; however, it expands into the following five lines of code:

```
if ($age < 18) {
    $age_description = "child";
} else {
    $age_description = "adult";
}
```

The ternary operator is so named because it has three operands: a condition to check (`$age < 18` in the previous code), a result if the condition is `true` ("child"), and a result if the condition is `false` ("adult"). Although we hope you never have to use the ternary operator, it is at least important to know how it works in case you stumble across it.

The other special operator is the execution operator, which is the backtick symbol (```). The position of the backtick key varies depending on your keyboard, but it is likely to be just to the left of the `1` key (above `Tab`). The execution operator executes the program inside the backticks, returning any text the program outputs. Here is an example:

```
<?php
    $i = `ls -l`;
    echo $i;
?>
```

This executes the `ls` program, passing in `-l` (a lowercase *L*) to get the long format, and stores all its output in `$i`. You can make the command as long or as complex as you like, including piping to other programs. You can also use PHP variables inside the command.

Switching

Having multiple `if` statements in one place is ugly, slow, and prone to errors. Consider the code in Listing 44.3.

LISTING 44.3 How Multiple Conditional Statements Lead to Ugly Code

```
<?php
    $cat_age = 3;

    if ($cat_age == 1) {
        echo "Cat age is 1";
    } else {
        if ($cat_age == 2) {
            echo "Cat age is 2";
        } else {
            if ($cat_age == 3) {
                echo "Cat age is 3";
            } else {
                if ($cat_age == 4) {
                    echo "Cat age is 4";
                } else {
                    echo "Cat age is unknown";
                }
            }
        }
    }
}
?>
```

Even though it certainly works, the code in Listing 44.3 is a poor solution to the problem. Much better is a `switch/case` block, which transforms the previous code into what's shown in Listing 44.4.

LISTING 44.4 Using a `switch/case` Block

```
<?php
    $cat_age = 3;

    switch ($cat_age) {
        case 1:
            echo "Cat age is 1";
            break;
        case 2:
            echo "Cat age is 2";
            break;
        case 3:
            echo "Cat age is 3";
            break;
```

```

case 4:
    echo "Cat age is 4";
    break;
default:
    echo "Cat age is unknown";
}
?>

```

Although it is only slightly shorter, Listing 44.4 is a great deal more readable and much easier to maintain. A `switch/case` group is made up of a `switch()` statement in which you provide the variable you want to check, followed by numerous `case` statements. Notice the `break` statement at the end of each `case`. Without that, PHP would execute each `case` statement beneath the one it matches. Calling `break` causes PHP to exit the `switch/case`. Notice also that there is a default `case` at the end that catches everything that has no matching case.

It is important that you not use `case default:` but merely `default:`. Also, it is the last `case` label, so it has no need for a `break` statement because PHP exits the `switch/case` block there anyway.

Loops

PHP has four ways you can execute a block of code multiple times: by using `while`, `for`, `foreach`, or `do...while`. Of the four, only `do...while` sees little use; the others are popular, and you will certainly encounter them in other people's scripts.

The most basic loop is the `while` loop, which executes a block of code for as long as a given condition is `true`. So, you can write an infinite loop—a block of code that continues forever—with this PHP:

```

<?php
    $i = 10;
    while ($i >= 10) {
        $i += 1;
        echo $i;
    }
?>

```

The loop block checks whether `$i` is greater or equal to 10 and, if that condition is `true`, adds 1 to `$i` and prints it. Then it goes back to the loop condition again. Because `$i` starts at 10 and you only ever add numbers to it, that loop continues forever. With two small changes, you can make the loop count down from 10 to 0:

```

<?php
    $i = 10;
    while ($i >= 0) {
        $i -= 1;
        echo $i;
    }

```

```
}  
?>
```

So, this time you check whether `$i` is greater than or equal to 0 and subtract 1 from it with each loop iteration. You typically use `while` loops when you are unsure how many times the code needs to loop because `while` keeps looping until an external factor stops it.

With a `for` loop, you specify precise limits on its operation by giving it a declaration, a condition, and an action. That is, you specify one or more variables that should be set when the loop first runs (the *declaration*), you set the circumstances that will cause the loop to terminate (the *condition*), and you tell PHP what it should change with each loop iteration (the *action*). That last part is what really sets a `for` loop apart from a `while` loop: You usually tell PHP to change the condition variable with each iteration.

You can rewrite the script that counts down from 10 to 0 by using a `for` loop:

```
<?php  
for($i = 10; $i >= 0; $i -- 1) {  
    echo $i;  
}  
?>
```

This time you do not need to specify the initial value for `$i` outside the loop, and you also do not need to change `$i` inside the loop; it is all part of the `for` statement. The actual amount of code is really the same, but for this purpose, the `for` loop is arguably tidier and therefore easier to read. With the `while` loop, the `$i` variable was declared outside the loop and so was not explicitly attached to the loop.

The third loop type is `foreach`, which is specifically for arrays and objects, although it is rarely used for anything other than arrays. A `foreach` loop iterates through each element in an array (or each variable in an object), optionally providing both the key name and the value.

In its simplest form, a `foreach` loop looks like this:

```
<?php  
foreach($myarr as $value) {  
    echo $value;  
}  
?>
```

This loops through the `$myarr` array you created earlier, placing each value in the `$value` variable. You can modify it so you get the keys as well as the values from the array, like this:

```
<?php  
foreach($myarr as $key => $value) {  
    echo "$key is set to $value\n";  
}  
?>
```

As you can guess, this time the array keys go in `$key`, and the array values go in `$value`. One important characteristic of the `foreach` loop is that it goes from the start of the array to the end and then stops—and by *start* we mean the first item to be added rather than the lowest index number. This script shows this behavior:

```
<?php
$array = array(6 => "Hello", 4 => "World",
              2 => "Wom", 0 => "Bat");
foreach($array as $key => $value) {
    echo "$key is set to $value\n";
}
?>
```

If you try this script, you will see that `foreach` prints the array in the original order of 6, 4, 2, 0 rather than the numerical order of 0, 2, 4, 6.

The `do...while` loop works like the `while` loop, with the exception that the condition appears at the end of the code block. This small syntactical difference means a lot, though, because a `do...while` loop is always executed at least once. Consider this script:

```
<?php
$i = 10;
do {
    $i -= 1;
    echo $i;
} while ($i < 10);
?>
```

Without running the script, what do you think it will do? One possibility is that it will do nothing; `$i` is set to 10, and the condition states that the code must loop only while `$i` is less than 10. However, a `do...while` loop always executes once, so what happens is that `$i` is set to 10, and PHP enters the loop, decrements `$i`, prints it, and then checks the condition for the first time. At this point, `$i` is indeed less than 10, so the code loops, `$i` is decremented again, the condition is rechecked, `$i` is decremented again, and so on. This is, in fact, an infinite loop and so should be avoided!

If you ever want to exit a loop before it has finished, you can use the same `break` statement that you used earlier to exit a `switch/case` block. This becomes more interesting if you find yourself with *nested* loops (loops inside loops). This is a common situation. For example, you might want to loop through all the rows in a chessboard and, for each row, loop through each column. Calling `break` exits only one loop or `switch/case`, but you can use `break 2` to exit two loops or `switch/cases`, or `break 3` to exit three, and so on.

Including Other Files

Unless you are restricting yourself to the simplest programming ventures, you will want to share code among your scripts at some point. The most basic need for this is to have a standard header and footer for your website, with only the body content changing.

However, you might also find yourself with a small set of custom functions you use frequently, and it would be an incredibly bad move to simply copy and paste the functions into each of the scripts that use them.

The most common way to include other files is by using the `include` keyword. Save this script as `include1.php`:

```
<?php
    for($i = 10; $i >= 0; $i -= 1) {
        include "echo_i.php";
    }
?>
```

Then save this script as `echo_i.php`:

```
<?php
    echo $i;
?>
```

If you run `include1.php`, PHP loops from 10 to 0 and includes `echo_i.php` each time. For its part, `echo_i.php` just prints the value of `$i`, which is a crazy way of performing an otherwise simple operation, but it does demonstrate how included files share data. Note that the `include` keyword in `include1.php` is inside a PHP block, but you reopen PHP inside `echo_i.php`. This is important because PHP exits PHP mode for each new file, so you always have a consistent entry point.

Basic Functions

PHP has a vast number of built-in functions that enable you to manipulate strings, connect to databases, and more. There is not room here to cover even 10 percent of the functions. For more detailed coverage of functions, check the “References” section at the end of this chapter.

Strings

Several important functions are used for working with strings, and there are many more of them that are less frequently used that we do not cover here. We look at the most important functions here, ordered by difficulty—easiest first!

The easiest function is `strlen()`, which takes a string as its parameter and returns the number of characters in there, like this:

```
<?php
    $ourstring = " The Quick Brown Box Jumped Over The Lazy Dog ";
    echo strlen($ourstring);
?>
```

We use this same string in subsequent examples to save space. If you execute this script, it outputs `48` because 48 characters are in the string. Note the 2 spaces on either side of

the text, which pad the 44-character phrase up to 48 characters. You can fix that padding with the `trim()` function, which takes a string to trim and returns it with all the whitespace removed from either side. This is a commonly used function because many strings have an extra new line at the end or a space at the beginning, and `trim()` cleans them up perfectly.

Using `trim()`, you can turn the 48-character string into a 44-character string (the same thing but without the extra spaces) like this:

```
echo trim($ourstring);
```

Keep in mind that `trim()` returns the trimmed string, so it outputs "The Quick Brown Box Jumped Over The Lazy Dog". You can modify it so `trim()` passes its return value to `strlen()` so that the code trims it and then outputs its trimmed length:

```
echo strlen(trim($ourstring));
```

PHP always executes the innermost functions first, so the previous code passes `$ourstring` through `trim()`, uses the return value of `trim()` as the parameter for `strlen()`, and prints it.

Of course, everyone knows that boxes do not jump over dogs; the usual phrase is "the quick brown fox." Fortunately, there is a function to fix that problem: `str_replace()`. Note that it has an underscore in it. (PHP is inconsistent on this matter, so you really need to memorize the function names.)

The `str_replace()` function takes three parameters: the text to search for, the text to replace it with, and the string you want to work with. When working with search functions, people often talk about *needles* and *haystacks*. In this situation, the first parameter is the needle (the thing to find), and the third parameter is the haystack (what you are searching through).

So, you can fix the error and correct *box* to *fox* with this code:

```
echo str_replace("Box", "Fox", $ourstring);
```

There are two little addendums to make here. First, note that we have specified "Box" as opposed to "box" because that is how it appears in the text. The `str_replace()` function is a *case-sensitive* function, which means it does not consider "Box" to be the same as "box". If you want to do a non-case-sensitive search and replace, you can use the `stri_replace()` function, which works the same way but doesn't care about case.

The second addendum is that because you are actually changing only one character (*B* to *F*), you need not use a function at all. PHP enables you to read (and change) individual characters of a string by specifying the character position inside braces (`{}`). As with arrays, strings are zero based, which means in the `$ourstring` variable `$ourstring{0}` is T, `$ourstring{1}` is h, `$ourstring{2}` is e, and so on. You could use this instead of `str_replace()`, like this:

```
<?php
```

```
$ourstring = " The Quick Brown Box Jumped Over The Lazy Dog ";
$ourstring{18} = "F";
echo $ourstring;
?>
```

You can extract part of a string by using the `substr()` function, which takes a string as its first parameter, a start position as its second parameter, and an optional length as its third parameter. Optional parameters are common in PHP. If you do not provide them, PHP assumes a default value. In this case, if you specify only the first two parameters, PHP copies from the start position to the end of the string. If you specify the third parameter, PHP copies that many characters from the start. You can write a simple script to print "Lazy Dog" by setting the start position to 38, which, remembering that PHP starts counting string positions from 0, copies from the 39th character to the end of the string:

```
echo substr($ourstring, 38);
```

If you just want to print the word "Lazy", you need to use the optional third parameter to specify the length as 4, like this:

```
echo substr($ourstring, 38, 4);
```

You can also use the `substr()` function with negative second and third parameters. If you specify just the first and second parameters and provide a negative number for the second parameter, `substr()` counts backward from the end of the string. So, rather than specifying 38 for the second parameter, you can use -10, so it takes the last 10 characters from the string. Using a negative second parameter and positive third parameter counts backward from the end string and then uses a forward length. You can print "Lazy" by counting 10 characters back from the end and then taking the next 4 characters forward:

```
echo substr($ourstring, -10, 4);
```

Finally, you can use a negative third parameter, too, which also counts back from the end of the string. For example, using "-4" as the third parameter means to take everything except the last four characters. Confused yet? This code example should make it clear:

```
echo substr($ourstring, -19, -11);
```

This counts 19 characters backward from the end of the string (which places it at the `o` in `Over`) and then copies everything from there until 11 characters before the end of the string. That prints `Over The`. You could write the same thing using -19 and 8, or even 29 and 8; there is more than one way to do it.

Moving on, the `strpos()` function returns the position of a particular substring inside a string; however, it is most commonly used to answer the question "Does this string contain a specific substring?" You need to pass two parameters to it: a haystack and a needle. (Yes, that's a different order from `str_replace()`.)

In its most basic use, `strpos()` can find the first instance of `Box` in your phrase, like this:

```
echo strpos($ourstring, "Box");
```

This outputs 18 because that is where the `B` in `Box` starts. If `strpos()` cannot find the substring in the parent string, it returns `false` rather than the position. Much more helpful, though, is the ability to check whether a string contains a substring; a first attempt to check whether your string contains the word `The` might look like this:

```
<?php
    $ourstring = "The Quick Brown Box Jumped Over The Lazy Dog";
    if (strpos($ourstring, "The")) {
        echo "Found 'The'!\n";
    } else {
        echo "'The' not found!\n";
    }
?>
```

Note that we have temporarily taken out the leading and trailing white space from `$ourstring` and are using the return value of `strpos()` for the conditional statement. This reads, “If the string is found, print a message; if not, print another message.” Or does it?

Run the script, and you see that it prints the “not found” message. The reason for this is that `strpos()` returns `false` if the substring is not found and otherwise returns the position where it starts. Recall that any nonzero number equates to `true` in PHP, which means that `0` equates to `false`. With that in mind, what is the string index of the first `The` in your phrase? Because PHP’s strings are zero based, and you no longer have the spaces on either side of the string, the `The` is at position `0`, which your conditional statement evaluates to `false` (hence, the problem).

The solution here is to check for identity. You know that `0` and `false` are equal, but they are not identical because `0` is an integer, whereas `false` is a Boolean. So, you need to rewrite the conditional statement to see whether the return value from `strpos()` is identical to `false`, and if it is, the substring was not found:

```
<?php
    $ourstring = "The Quick Brown Box Jumped Over The Lazy Dog";
    if (strpos($ourstring, "The") !== false) {
        echo "Found 'The'!\n";
    } else {
        echo "'The' not found!\n";
    }
?>
```

Arrays

Working with arrays is no easy task, but PHP makes it less difficult by providing a selection of functions that can sort, shuffle, intersect, and filter them. (As with other functions, there is only space here to cover a selection; this chapter is by no means a definitive reference to PHP’s array functions.)

The easiest array function to use is `array_unique()`, which takes an array as its only parameter and returns the same array with all duplicate values removed. Also in the realm of “so easy you do not need a code example” is the `shuffle()` function, which takes an array as its parameter and randomizes the order of its elements. Note that `shuffle()` does not return the randomized array; it uses your parameter as a reference and scrambles it directly. The last too-easy-to-demonstrate function is `in_array()`, which takes a value as its first parameter and an array as its second and returns `true` if the value is in the array.

With those functions out of the way, we can focus on the more interesting functions, two of which are `array_keys()` and `array_values()`. They both take an array as their only parameter and return a new array made up of the keys in the array or the values of the array, respectively. Using the `array_values()` function is an easy way to create a new array of the same data, just without the keys. This is often used if you have numbered your array keys, deleted several elements, and want to reorder the array.

The `array_keys()` function creates a new array where the values are the keys from the old array, like this:

```
<?php
    $myarr = array("foo" => "red", "bar" => "blue", "baz" => "green");
    $mykeys = array_keys($myarr);
    foreach($mykeys as $key => $value) {
        echo "$key = $value\n";
    }
?>
```

This prints "0 = foo", "1 = bar", and "2 = baz".

Several functions are used specifically for array sorting, but only two get much use: `asort()` and `ksort()`. `asort()` sorts an array by its values, and `ksort()` sorts the array by its keys. Given the array `$myarr` from the previous example, sorting by values would produce an array with elements in the order `bar/blue`, `baz/green`, and `foo/red`. Sorting by key would give the elements in the order `bar/blue`, `baz/green`, and `foo/red`. As with the `shuffle()` function, both `asort()` and `ksort()` do their work *in place*, meaning they return no value but directly alter the parameter you pass in. For interest's sake, you can also use `arsort()` and `krsort()` for reverse value sorting and reverse key sorting, respectively.

This code example reverse sorts the array by value and then prints it, as before:

```
<?php
    $myarr = array("foo" => "red", "bar" => "blue", "baz" => "green");
    arsort($myarr);
    foreach($myarr as $key => $value) {
        echo "$key = $value\n";
    }
?>
```

Previously, when discussing constants, we mentioned the `extract()` function, which converts an array into individual variables; now it is time to start using it for real. You need to provide three variables: the array you want to extract, how you want the variables prefixed, and the prefix you want used. Technically, the last two parameters are optional, but practically you should always use them to properly namespace your variables and keep them organized.

The second parameter must be one of the following:

- ▶ **EXTR_OVERWRITE**—If the variable exists already, overwrites it.
- ▶ **EXTR_SKIP**—If the variable exists already, skips it and moves on to the next variable.
- ▶ **EXTR_PREFIX_SAME**—If the variable exists already, uses the prefix specified in the third parameter.
- ▶ **EXTR_PREFIX_ALL**—Prefixes all variables with the prefix in the third parameter, regardless of whether it exists already.
- ▶ **EXTR_PREFIX_INVALID**—Uses a prefix only if the variable name would be invalid (for example, starting with a number).
- ▶ **EXTR_IF_EXISTS**—Extracts only variables that already exist. We have never seen this used.

You can also, optionally, use the bitwise OR operator, `|`, to add in `EXTR_REFS` to have `extract()` use references for the extracted variables. In general use, `EXTR_PREFIX_ALL` is preferred because it guarantees namespacing. `EXTR_REFS` is required only if you need to be able to change the variables and have those changes reflected in the array.

This next script uses `extract()` to convert `$myarr` into individual variables, `$arr_foo`, `$arr_bar`, and `$arr_baz`:

```
<?php
    $myarr = array("foo" => "red", "bar" => "blue", "baz" => "green");
    extract($myarr, EXTR_PREFIX_ALL, 'arr');
?>
```

Note that the array keys are "foo", "bar", and "baz" and that the prefix is "arr", but that the final variables will be `$arr_foo`, `$arr_bar`, and `$arr_baz`. PHP inserts an underscore between the prefix and array key.

Files

As you have learned elsewhere in the book, the UNIX philosophy is that everything is a file. In PHP, this is also the case: A selection of basic file functions is suitable for opening and manipulating files, but those same functions can also be used for opening and manipulating network sockets. We cover both here.

Two basic read and write functions for files make performing these basic operations easy. They are `file_get_contents()`, which takes a filename as its only parameter and returns

the file's contents as a string, and `file_put_contents()`, which takes a filename as its first parameter and the data to write as its second parameter.

Using these two functions, you can write a script that reads all the text from one file, `filea.txt`, and writes it to another, `fileb.txt`:

```
<?php
    $text = file_get_contents("filea.txt");
    file_put_contents("fileb.txt", $text);
?>
```

Because PHP enables you to treat network sockets like files, you can also use `file_get_contents()` to read text from a website, like this:

```
<?php
    $text = file_get_contents("http://www.slashdot.org");
    file_put_contents("fileb.txt", $text);
?>
```

The problem with using `file_get_contents()` is that it loads the whole file into memory at once; that's not practical if you have large files or even smaller files being accessed by many users. An alternative is to load the file piece by piece, which can be accomplished by using the following five functions: `fopen()`, `fclose()`, `fread()`, `fwrite()`, and `feof()`. The *f* in these function names stands for *file*, so they open, close, read from, and write to files and sockets. The last function, `feof()`, returns `true` if the end of the file has been reached.

On the surface, the `fopen()` function looks straightforward, though it takes a bit of learning to use properly. Its first parameter is the filename you want to open, which is easy enough. However, the second parameter is where you specify how you want to work with the file, and you should specify one of the following:

- ▶ **r**—Read-only; it overwrites the file.
- ▶ **r+**—Reading and writing; it overwrites the file.
- ▶ **w**—Write-only; it erases the existing contents and overwrites the file.
- ▶ **w+**—Reading and writing; it erases the existing content and overwrites the file.
- ▶ **a**—Write-only; it appends to the file.
- ▶ **a+**—Reading and writing; it appends to the file.
- ▶ **x**—Write-only, but only if the file does not exist.
- ▶ **a+**—Reading and writing, but only if the file does not exist.

Optionally, you can also add `b` (for example, `a+b` or `rb`) to switch to binary mode. This is recommended if you want your scripts and the files they write to work smoothly on other platforms.

When you call `fopen()`, you should store the return value. It is a resource known as a *file handle*, which the other file functions all need to do their jobs. The `fread()` function, for example, takes the file handle as its first parameter and the number of bytes to read as its second, and it returns the content in its return value. The `fclose()` function takes the file handle as its only parameter and frees up the file.

So, you can write a simple loop to open a file, read it piece by piece, print the pieces, and then close the handle:

```
<?php
    $file = fopen("filea.txt", "rb");
    while (!feof($file)) {
        $content = fread($file, 1024);
        echo $content;
    }
    fclose($file);
?>
```

This leaves only the `fwrite()` function, which takes the file handle as its first parameter and the string to write as its second. You can also provide an integer as the third parameter, specifying the number of bytes you want to write of the string, but if you exclude this, `fwrite()` writes the entire string.

Recall that you can use `a` as the second parameter to `fopen()` to append data to a file. So, you can combine that with `fwrite()` to have a script that adds a line of text to a file each time it is executed:

```
<?php
    $file = fopen("filea.txt", "ab");
    fwrite($file, "Testing\n");
    fclose($file);
?>
```

To make this script a little more exciting, you can stir in a new function, `filesize()`, that takes a filename (not a file handle, but an actual filename string) as its only parameter and returns the file's size, in bytes. Using the `filesize()` function brings the script to this:

```
<?php
    $file = fopen("filea.txt", "ab");
    fwrite($file, "The filesize was" . filesize("filea.txt") . "\n");
    fclose($file);
?>
```

Although PHP automatically cleans up file handles for you, it is still best to use `fclose()` yourself so that you are always in control.

Miscellaneous

Several functions do not fall under the other categories and so are covered here. The first one is `isset()`, which takes one or more variables as its parameters and returns `true` if they have been set. It is important to note that a variable with a value set to something that would be evaluated to `false`—such as `0` or an empty string—still returns `true` from `isset()` because this function does not check the value of the variable. It merely checks that it is set; hence, the name.

The `unset()` function also takes one or more variables as its parameters, simply deleting the variable(s) and freeing up the memory. With `isset()` and `unset()`, you can write a script that checks for the existence of a variable and, if it exists, deletes it (see Listing 44.5).

LISTING 44.5 Setting and Unsetting Variables

```
<?php
$name = "Ildiko";
if (isset($name)) {
    echo "Name was set to $name\n";
    unset($name);
} else {
    echo "Name was not set";
}

if (isset($name)) {
    echo "Name was set to $name\n";
    unset($name);
} else {
    echo "Name was not set";
}
?>
```

This script runs the same `isset()` check twice, but it uses `unset()` on the variable after the first check. It therefore prints "Name was set to Ildiko" and then "Name was not set".

Perhaps the most frequently used function in PHP is `exit`, although purists will tell you that it is, in fact, a language construct rather than a function. `exit` terminates the processing of the script as soon as it is executed, which means subsequent lines of code are not executed. That is really all there is to it; it barely deserves an example, but here is one just to make sure:

```
<?php
exit;
echo "Exit is a language construct!\n";
?>
```

This script prints nothing because the `exit` comes before the `echo`.

One function we can guarantee you will use a lot is `var_dump()`, which dumps out information about a variable, including its value, to the screen. This function is invaluable for arrays because it prints every value and, if one or more of the elements is an array, it prints all the elements from those, and so on. To use this function, just pass a variable to it as its only parameter, as shown here:

```
<?php
    $drones = array("Graham", "Julian", "Nick", "Paul");
    var_dump($drones);
?>
```

The output from this script looks as follows:

```
array(4) {
    [0]=>
    string(6) "Graham"
    [1]=>
    string(6) "Julian"
    [2]=>
    string(4) "Nick"
    [3]=>
    string(4) "Paul"
}
```

The `var_dump()` function sees a lot of use as a basic debugging technique because using it is the easiest way to print variable data to the screen to verify it.

Finally, we briefly discuss regular expressions—with the emphasis on *briefly* because regular expression syntax is covered elsewhere in this book, and the only unique thing relevant to PHP is the functions you use to run the expressions. You have the choice of either *Perl-Compatible Regular Expressions (PCRE)* or POSIX Extended regular expressions, but there really is little to choose between them in terms of functionality offered. For this chapter, we use the PCRE expressions because, to the best of our knowledge, they see more use by other PHP programmers.

The main PCRE functions are `preg_match()`, `preg_match_all()`, `preg_replace()`, and `preg_split()`. We start with `preg_match()` because it provides the most basic functionality, returning `true` if one string matches a regular expression. The first parameter to `preg_match()` is the regular expression you want to search for, and the second is the string to match. So, if you want to check whether a string has the word `Best`, `Test`, `rest`, `zest`, or any other word containing `est` preceded by any letter of either case, you could use this PHP code:

```
$result = preg_match("/[A-Za-z]est/", "This is a test");
```

Because the test string matches the expression, `$result` is set to `1` (`true`). If you change the string to a nonmatching result, you get `0` as the return value.

The next function is `preg_match_all()`, which gives you an array of all the matches it found. However, to be most useful, it takes the array to fill with matches as a by reference parameter and saves its return value for the number of matches that were found.

We suggest that you use `preg_match_all()` and `var_dump()` to get a feel for how the `preg_match_all()` function works. This example is a good place to start:

```
<?php
    $string = "This is the best test in the west";
    $result = preg_match_all("/[A-Za-z]est/", $string, $matches);
    var_dump($matches);
?>
```

It outputs the following:

```
array(1) {
    [0]=>
    array(3) {
        [0]=>
        string(4) "best"
        [1]=>
        string(4) "test"
        [2]=>
        string(4) "west"
    }
}
```

Notice that the `$matches` array is actually multidimensional in that it contains one element, which itself is an array containing all the matches to the regular expression. This is because the expression has no *subexpressions*, meaning no independent matches using parentheses. If you had subexpressions, each would have its own element in the `$matches` array, containing its own array of matches.

Moving on, `preg_replace()` is used to change all substrings that match a regular expression into something else. The basic manner of using this is quite easy: You search for something with a regular expression and provide a replacement for it. However, a more useful variant is *back referencing*, using the match as part of the replacement. For the sake of this example, say that you have written a tutorial on PHP but want to process the text so each reference to a function is followed by a link to the PHP manual.

PHP manual page URLs take the form `www.php.net/<somefunc>` (for example, `www.php.net/preg_replace`). The string you need to match is a function name, which is a string of alphabetic characters, potentially also mixed with numbers and underscores and terminated with parentheses, `()`. As a replacement, you can use the match you found, surrounded in HTML emphasis tags (``), and then a link to the relevant PHP manual page. Here is how all this looks in code:

```
<?php
    $regex = "/([A-Za-z0-9_]*)(\()/";
```

```

$replace = "<em>$1</em> (<a href=\"http://www.php.net/$1\">>manual</A>)";
$haystack = "File_get_contents() is easier than using fopen().";
$result = preg_replace($regex, $replace, $haystack);
echo $result;
?>

```

The `$1` is the back reference; it will be substituted with the results from the first subexpression. The way we have written the regular expression is very exact. The `[A-Za-z0-9_]*` part, which matches the function name, is marked as a subexpression. After that is `\` (`\`), which means the exact symbols (and), not the regular expression meanings of them, which means that `$1` in the replacement will contain `fopen` rather than `fopen()`, which is how it should be. Of course, anything that is not back referenced in the replacement is removed, so you have to put the `()` after the first `$1` (not in the hyperlink) to repair the function name.

After all that work, the output is perfect:

```

<em>File_get_contents()</em> (<a href="http://www.php.net/
file_get_contents">>manual</A>) is easier than using <em>fopen()
</em> (<a href="http://www.php.net/fopen">>manual</A>).

```

Handling HTML Forms

Given that PHP's primary role is handling web pages, you might wonder why this section has been left until so late in the chapter. It is because handling HTML forms is so central to PHP that it is essentially automatic.

Consider this form:

```

<form method="POST" action="thispage.php">
User ID: <input type="text" name="UserID" /><br />
Password: <input type="password" name="Password" /><br />
<input type="submit" />
</form>

```

When a visitor clicks Submit, `thispage.php` is called again, and this time PHP has the variables available to it inside the `$_REQUEST` array. Given that script, if the user enters 12345 and `frosties` as her user ID and password, PHP provides `$_REQUEST['UserID']` set to 12345 and `$_REQUEST['Password']` set to `frosties`. Note that it is important that you use HTTP POST unless you specifically want GET. POST enables you to send a great deal more data and stops people from tampering with your URL to try to find holes in your script.

Is that it? Well, almost. You now know how to retrieve user data, but you should be sure to sanitize it so users do not try to sneak HTML or JavaScript into your database as something you think is innocuous. PHP gives you the `strip_tags()` function for this purpose. It takes a string and returns the same string with all HTML tags removed.

Databases

The ease with which PHP can be used to create dynamic, database-driven websites is the key reason to use it for many people. The stock build of PHP comes with support for MySQL, PostgreSQL, SQLite, Oracle, Microsoft SQL Server, ODBC, plus several other popular databases, so you are sure to find something to work with your data.

If you want to, you can learn all the individual functions for connecting to and manipulating each database PHP supports, but a much smarter, or at least easier, idea is to use `PEAR::DB`, which is an abstraction layer over the databases that PHP supports. You write your code once, and—with the smallest of changes—it works on every database server.

PEAR is the script repository for PHP, and it contains numerous tools and prewritten solutions for common problems. `PEAR::DB` is perhaps the most popular part of the PEAR project, but it is worth checking out the PEAR site (<https://pear.php.net>) to see whether anything else catches your eye.

To get basic use out of `PEAR::DB`, you need to learn how to connect to a database, run an SQL query, and work with the results. This is not an SQL tutorial, so we have assumed that you are already familiar with the language. For the sake of this tutorial, we have also assumed that you are working with a database called `dentists` and a table called `patients` that contains the following fields:

- ▶ `ID`—The primary key, an auto-incrementing integer for storing a number unique to each patient
- ▶ `Name`—A `varchar(255)` field for storing a patient name
- ▶ `Age`—An integer
- ▶ `Sex`—1 for male, 2 for female
- ▶ `Occupation`—A `varchar(255)` field for storing a patient occupation

Also for the sake of this tutorial, we use a database server at IP address `10.0.0.1`, running MySQL, with username `ubuntu` and password `alm65z`. You need to replace these details with your own; use `localhost` for connecting to the local server.

The first step to using `PEAR::DB` is to include the standard `PEAR::DB` file, `DB.php`. Your PHP will be configured to look inside the PEAR directory for `include()` files, so you do not need to provide any directory information.

`PEAR::DB` is object oriented, and you specify your connection details at the same time as you create the initial DB object. This is done using a URL-like system that specifies the database server type, username, password, server, and database name all in one. After you have specified the database server here, everything else is abstracted, meaning you only need to change the connection line to port your code to another database server.

This script in Listing 44.6 connects to our server and prints a status message.

LISTING 44.6 Connecting to a Database Through PEAR::DB

```

<?php
    include("DB.php");
    $dsn = "mysql://ubuntu:alm65z@10.0.0.1/dentists";
    $conn = DB::connect($dsn);
    if (DB::isError($conn)) {
        echo $conn->getMessage() . "\n";
    } else {
        echo "Connected successfully!\n";
    }
?>

```

You should be able to see how the connection string breaks down. It is server name first, then a username and password separated by a colon, then an @ symbol followed by the IP address to which to connect, and then a slash and the database name. Notice how the call to connect is `DB::connect()`, which calls `PEAR::DB` directly and returns a database connection object for storage in `$conn`. The variable name `$dsn` was used for the connection details because it is a common acronym that stands for data source name.

If `DB::connect()` successfully connects to a server, it returns a database object you can use to run SQL queries. If not, you get an error returned that you can query using functions such as `getMessage()`. The script in Listing 44.6 prints the error message if the connection fails, but it also prints a message on success. Next, you change that so you run an SQL query if we have a connection.

Running SQL queries is done through the `query()` function of the database connection, passing in the SQL you want to execute. This then returns a query result that can be used to get the data. This query result can be thought of as a multidimensional array because it has many rows of data, each with many columns of attributes. This is extracted using the `fetchInto()` function, which loops through the query result, converting one row of data into an array that it sends back as its return value. You need to pass in two parameters to `fetchInto()` to specify where the data should be stored and how you want it stored. Unless you have unusual needs, specifying `DB_FETCHMODE_ASSOC` for the second parameter is a smart move.

Listing 44.7 shows the new script.

LISTING 44.7 Running a Query Through PEAR::DB

```

<?php
    include("DB.php");
    $dsn = "mysql://ubuntu:alm65z@10.0.0.1/dentists";
    $conn = DB::connect($dsn);
    if (DB::isError($conn)) {
        echo $conn->getMessage() . "\n";
    } else {

```

```
echo "Connected successfully!\n";
$result = $conn->query("SELECT ID, Name FROM patients;");
while ($result->fetchInto($row, DB_FETCHMODE_ASSOC)) {
    extract($row, EXTR_PREFIX_ALL, 'pat');
    echo "$pat_ID is $pat_Name\n";
}
}
?>
```

The first half of Listing 44.7 is identical to the script in Listing 44.6, with all the new action happening if a successful connection occurs.

Going along with the saying “never leave to PHP what you can clean up yourself,” the current script has problems. It does not clean up the query result, and it does not close the database connection. If this code were being used in a longer script that ran for several minutes, this would be a huge waste of resources. Fortunately, you can free up the memory associated with these two issues by calling `$result->free()` and `$conn->disconnect()`. If you add those two function calls to the end of the script, it is complete.

References

- ▶ **<https://secure.php.net>**—The best place to look for information is the PHP online manual. It is comprehensive, well written, and updated regularly.
- ▶ **www.phpbuilder.com**—This is a large PHP scripts and tutorials site where you can learn new techniques and also chat with other PHP developers.
- ▶ **www.zend.com**—This is the home page of a company founded by two of the key developers of PHP. Zend develops and sells proprietary software, including a powerful IDE and a code cache, to aid PHP developers.
- ▶ **<https://pear.php.net>**—The home of the PEAR project contains a large collection of software you can download and try, and it has thorough documentation for it all.
- ▶ **www.phparch.com**—Quite a few good PHP magazines are around, but *PHP Architect* leads the way. It posts some of its articles online for free, and its forums are good, too.
- ▶ ***PHP and MySQL Web Development* by Luke Welling and Laura Thomson**—This book is the best choice for beginning developers.
- ▶ ***PHP in a Nutshell* by Paul Hudson**—This concise, to-the-point book covers all aspects of PHP.
- ▶ ***Advanced PHP Programming* by George Schlossnagle**—As its title says, this book is for advanced developers.