

EXPERT INSIGHT

Mastering Ubuntu Server

Explore the versatile, powerful Linux Server distribution
Ubuntu 22.04 with this comprehensive guide

Fourth Edition



Jay LaCroix

<packt>

Mastering Ubuntu Server

Fourth Edition

Explore the versatile, powerful Linux Server distribution Ubuntu 22.04 with this comprehensive guide

Jay LaCroix



BIRMINGHAM—MUMBAI

Mastering Ubuntu Server

Fourth Edition

Copyright © 2022 Packt Publishing

All rights reserved. No part of this book may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, without the prior written permission of the publisher, except in the case of brief quotations embedded in critical articles or reviews.

Every effort has been made in the preparation of this book to ensure the accuracy of the information presented. However, the information contained in this book is sold without warranty, either express or implied. Neither the author, nor Packt Publishing or its dealers and distributors, will be held liable for any damages caused or alleged to have been caused directly or indirectly by this book.

Packt Publishing has endeavored to provide trademark information about all of the companies and products mentioned in this book by the appropriate use of capitals. However, Packt Publishing cannot guarantee the accuracy of this information.

Senior Publishing Product Manager: Aaron Tanna

Acquisition Editor – Peer Reviews: Saby Dsilva

Project Editor: Namrata Katare

Content Development Editor: Liam Thomas Draper

Copy Editor: Safis Editing

Technical Editor: Aditya Sawant

Proofreader: Safis Editing

Indexer: Manju Arasan

Presentation Designer: Ganesh Bhadwalkar

First published: July 2016

Second edition: May 2018

Third edition: December 2020

Fourth edition: September 2022

Production reference: 1140922

Published by Packt Publishing Ltd.

Livery Place

35 Livery Street

Birmingham

B3 2PB, UK.

ISBN 978-1-80323-424-3

www.packt.com

Contributors

About the author

Based in Michigan, **Jay LaCroix** is an educator and technologist, specializing in Linux. After a successful tech career, Jay recently stepped away from corporate IT in order to launch his own company, LearnLinuxTV. There, Jay has produced hundreds of Linux-related training videos that are viewed by a half million people per month on average. In addition, Jay also writes books on Linux, with this one being a recent example.

Outside of technology, Jay also enjoys retro-gaming and martial arts.

I would like to thank all of my readers and subscribers for making all of this possible. It's an amazing honor that you've chosen me to lead you on your Linux journey! Also thanks to Tom Lawrence for encouraging me and giving me lots of advice. Last (but by no means least), thanks to my partner Alan, my sons, and all of my family and friends.

About the reviewers

Paride Legovini is a core Ubuntu developer, a Debian developer, and currently an Ubuntu Server software engineer at Canonical. He does packaging, Quality Assurance, and maintains the testing and continuous integration infrastructure of team-developed projects, like cloud-init and the Ubuntu Advantage tools, which includes automated integration testing done on all the major clouds. He has also joined the release engineering team for all the new Ubuntu releases since 2019, being responsible for the quality of Ubuntu Server.

His professional past is in academia, with an MSc in Physics, a Ph.D. in Geophysics, and his professional roles often been at the intersection between science and IT in different research institutions.

He spent 2014 in Antarctica, where he was in charge of maintenance and data acquisition, storage, backup, and transmission for several international scientific projects at the Concordia research station during the polar winter. The role leveraged Paride's all-around tinkerer nature, sometimes requiring operating a power tool, a soldering iron, and a Unix shell, not necessarily in that order.

I would like to thank the Debian project for giving the world a universal operating system for anybody to run, study, modify, and share. Debian realized the promise that free software would empower people, and has had a significant impact on my life and career. Thanks also to Canonical for popularizing GNU/Linux and for allowing me to participate in this review, to Jay LaCroix for turning what is clearly genuine passion into such an enjoyable book, and to Packt for inviting me to be a reviewer.

Mario Splivalo's amazement with computers started back when Commodore 64 dominated the "user-space." He took his first steps using BASIC on his dad's Commodore 64 before quickly shifting to assembler. He gradually moved to PCs where he discovered a great love for programming, systems design, and database administration. He switched to Linux (Knoppix, then Ubuntu) in the early 2000s and never looked back. He continued as a database administrator, programmer, and system administrator.

Mario spent five years with Canonical working mainly as an OpenStack engineer. It was a great experience that helped Mario to see how Ubuntu is created "from the inside", and participate in its development.

Currently, Mario works as a consultant dealing with databases extending into modern cloud-based architectures.

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



Table of Contents

Preface	xi
<hr/>	
Chapter 1: Deploying Ubuntu Server	1
<hr/>	
Technical requirements	2
Determining your server's role	2
Choosing a device for our server	3
Physical server • 3	
Physical desktop • 4	
Laptop • 4	
Virtual machine • 5	
Virtual private server • 5	
Raspberry Pi • 5	
Obtaining installation media	6
Creating a bootable flash drive	8
Planning the partitioning layout • 11	
Installing Ubuntu Server	12
Installing Ubuntu on a Raspberry Pi	25
Summary	28
Relevant tutorials	29
Chapter 2: Managing Users and Permissions	31
<hr/>	
Understanding the purpose of users and groups	32
Understanding when to use root	32
Using sudo to run privileged commands • 33	
Creating and removing users	34
Using useradd • 34	
Using adduser • 36	

Removing users • 37	
Understanding the /etc/passwd and /etc/shadow files	39
Understanding the /etc/passwd file • 39	
Understanding the /etc/shadow file • 41	
Distributing default configuration files with /etc/skel	43
Switching users	45
Managing groups	46
Managing passwords and password policies	49
Locking and unlocking user accounts • 49	
Setting password expiration information • 50	
Setting a password policy • 51	
Configuring administrator access with sudo	53
Setting permissions on files and directories	56
Viewing permissions • 56	
Changing permissions • 59	
Changing the ownership of objects • 60	
Summary	61
Relevant videos	61
Further reading	62
Chapter 3: Managing Software Packages	63
<hr/>	
Understanding Linux package management	63
Understanding the differences between Debian and Snap packages	65
Debian packages • 65	
Snap packages • 66	
Installing and removing software	68
Managing Debian packages with apt • 68	
Managing Snap packages with snap • 71	
Searching for packages	73
Managing package repositories	75
Adding additional repositories • 75	
Adding Personal Package Archives • 77	
Backing up and restoring Debian packages	78
Cleaning up orphaned apt packages	79
Taking advantage of hardware enablement updates	81
Summary	83
Relevant video	83

Further reading	83
Chapter 4: Navigating and Essential Commands	85
Learning essential Linux commands	85
Understanding the Linux filesystem layout	89
Viewing the contents of files	91
Viewing application log files	94
Summary	95
Relevant video	95
Further reading	95
Chapter 5: Managing Files and Directories	97
Copying, moving, and renaming files and directories	97
Editing files with the Nano and Vim text editors	99
Editing with Nano • 100	
Editing with Vim • 102	
Input and output streams	108
Using symbolic and hard links	111
Summary	113
Relevant videos	113
Chapter 6: Boosting Your Command-line Efficiency	115
Understanding the Linux shell	115
Understanding Bash history	117
Learning some useful command-line tricks	119
Understanding variables	122
Writing simple scripts	124
Putting it all together – Writing an rsync backup script	129
Summary	131
Relevant videos	131
Further reading	131
Chapter 7: Controlling and Managing Processes	133
Managing jobs	133
Understanding the ps command	136
Viewing running processes with ps • 136	

Configuring arguments to ps • 137	
Changing the priority of processes	141
Dealing with misbehaving processes	145
Managing system processes	148
Scheduling tasks with cron	151
Summary	153
Relevant videos	154
Further reading	154
Chapter 8: Monitoring System Resources	155
<hr/>	
Viewing disk usage	155
Using df • 156	
Diving deeper into disk usage • 158	
Monitoring memory usage	161
Understanding server memory • 161	
Managing swap • 163	
Understanding load average	166
Viewing resource usage with htop	168
Summary	172
Relevant videos	172
Further reading	172
Chapter 9: Managing Storage Volumes	173
<hr/>	
Adding additional storage volumes	173
Formatting and partitioning storage devices	176
Creating a partition • 177	
Formatting partitions • 180	
Mounting and unmounting volumes	182
Understanding the /etc/fstab file	184
Analyzing the contents of /etc/fstab • 184	
Adding to the /etc/fstab file • 186	
Backing up and restoring volumes	188
Utilizing LVM	190
Getting started with LVM • 191	
Formatting logical volumes • 195	
Removing volumes with LVM • 197	
Understanding LVM snapshots • 198	

Summary	199
Relevant videos	200
Further reading	200
Chapter 10: Connecting to Networks	201
Setting the hostname	201
Managing network interfaces	204
Assigning static IP addresses	209
Understanding Linux name resolution	213
Getting started with OpenSSH	215
Installing OpenSSH • 216	
Issuing commands with OpenSSH • 217	
Getting started with SSH key management	219
Generating public and private keys • 220	
Copying your public key to a remote server • 221	
Utilizing an SSH agent • 222	
Changing the passphrase of an OpenSSH key • 223	
Simplifying SSH connections with a config file	223
Summary	225
Relevant videos	225
Further reading	225
Chapter 11: Setting Up Network Services	227
Planning your IP address scheme	227
Setting up a DHCP server for serving IP addresses	231
Adding a DNS server	236
Setting up external DNS with bind • 237	
Setting up internal DNS and adding hosts • 239	
Setting up an internet gateway	243
Summary	245
Further reading	245
Chapter 12: Sharing and Transferring Files	247
File server considerations	247
Sharing files with Windows users via Samba	249
Setting up NFS shares	254

Transferring files with rsync	258
Transferring files with SCP	261
Summary	263
Relevant videos	263
Further reading	263
Chapter 13: Managing Databases	265
Preparations for setting up a database server	265
Installing MariaDB	267
Understanding the MariaDB configuration files	269
Managing MariaDB databases	272
Setting up a secondary database server	278
Summary	283
Further reading	283
Chapter 14: Serving Web Content	285
Installing and configuring Apache	285
Installing additional Apache modules	291
Securing Apache with TLS	293
Installing and configuring NGINX	299
Setting up and configuring Nextcloud	303
Summary	310
Relevant videos	310
Further reading	310
Chapter 15: Automating Server Configuration with Ansible	313
Understanding the need for configuration management	313
Why Ansible?	315
Creating a Git repository	316
Getting started with Ansible	319
Making your servers do your bidding	321
Setting up an inventory file and configuring Ansible settings •	321
Configuring client servers •	323
Putting it all together – automating web server deployment	327
Using Ansible’s pull method	330
Summary	334

Relevant videos	334
Further reading	334
Chapter 16: Virtualization	337
Prerequisites and considerations	337
Setting up a virtual machine server	339
Creating virtual machines	345
Bridging the virtual machine network	351
Simplifying virtual machine creation with cloning	354
Managing virtual machines via the command line	356
Summary	357
Relevant video	357
Further reading	357
Chapter 17: Running Containers	359
What is containerization?	359
Understanding the differences between Docker and LXD	361
Installing Docker	362
Managing Docker containers	363
Automating Docker image creation with Dockerfiles	371
Managing LXD containers	373
Summary	377
Relevant videos	377
Further reading	377
Chapter 18: Container Orchestration	379
Container orchestration	380
Preparing a lab environment for Kubernetes testing	381
Utilizing MicroK8s	383
Installing MicroK8s on Linux • 383	
Installing MicroK8s on macOS • 384	
Installing MicroK8s on Windows • 386	
Interacting with MicroK8s • 390	
Setting up a Kubernetes cluster	392
Preliminary setup • 393	
Installing Kubernetes • 397	

Deploying containers via Kubernetes	402
Summary	408
Relevant videos	408
Further reading	408
Chapter 19: Deploying Ubuntu in the Cloud	409
Understanding the difference between on-premises and cloud infrastructure	410
Important considerations when considering cloud computing as a potential solution	411
Becoming familiar with some basic AWS concepts	413
Creating an AWS account	416
Signing up for AWS • 417	
Implementing basic user security • 421	
Choosing a region	427
Deploying Ubuntu as an AWS EC2 instance	428
Setting up an IAM role for Session Manager • 428	
Creating an Ubuntu Server instance in AWS • 432	
Creating and deploying Ubuntu AMIs	444
Automatically scaling Ubuntu EC2 deployments with Auto Scaling	446
Creating a launch template • 447	
Creating an Auto Scaling group • 448	
Keeping costs down: understanding how to save money and make cost-effective decisions	455
Viewing billing information • 455	
Adding a billing alert • 456	
Removing unneeded backups • 456	
Running EC2 instances only when they're needed • 456	
Stopping or terminating unneeded EC2 instances • 457	
Taking the cloud further: additional resources to grow your knowledge	457
Online training and labs • 457	
Certification • 458	
Keep experimenting and learning • 458	
AWS documentation • 458	
Summary	458
Further reading	459
Chapter 20: Automating Cloud Deployments with Terraform	461
Why it's important to automate your infrastructure	462
Introduction to Terraform and how it can fit within your workflow	463

Installing Terraform	465
Automating an EC2 instance deployment	470
Managing security groups with Terraform	476
Using Terraform to destroy unused resources	479
Combining Ansible with Terraform for a full deployment solution	481
Summary	484

Chapter 21: Securing Your Server **485**

Lowering your attack surface	485
Understanding and responding to CVEs	490
Installing security updates	490
Automatically installing patches with the Canonical Livepatch service	494
Securing OpenSSH	495
Installing and configuring Fail2ban	498
MariaDB best practices for secure database servers	502
Setting up a firewall	505
Encrypting and decrypting disks with LUKS	507
Locking down sudo	509
Summary	510
Further reading	510

Chapter 22: Troubleshooting Ubuntu Servers **511**

Evaluating the scope	511
Conducting a root cause analysis	513
Viewing system logs	515
Tracing network issues	520
Troubleshooting resource issues	524
Diagnosing defective RAM	527
Summary	530
Further reading	530

Chapter 23: Preventing Disasters **531**

Preventing disasters	531
Utilizing Git for configuration management	533
Implementing a backup plan	538
Utilizing bootable recovery media	540

Summary	541
Further reading	542
Other Books You May Enjoy	545
<hr/>	
Index	549

Preface

Linux is an exciting platform. You can literally find it everywhere—desktops, laptops, phones, embedded appliances, and especially servers. Linux servers power many famous technologies around the world, and have a massive presence in physical and cloud server environments. Linux servers are scalable, stable, and powerful.

Way back in 1991, a computer science student in Finland, named Linus Torvalds, created a kernel as a school project. He created the Linux kernel. And little did he know at the time, Linux would see us through some of the most exciting innovations in infrastructure design. Things like multicore processors, virtualization, containerization, and cloud computing (to name a few) have transformed data centers all over the world. And Linux has been a part of all of those advancements.

Ubuntu, a popular distribution of Linux, has been a major driving force in the adoption of Linux. Ever since its first release back in 2004, Ubuntu has made Linux more approachable for newcomers, while simultaneously offering powerful features to server administrators. Ubuntu can be found on notebooks and workstations, in addition to servers.

Ubuntu Server enables administrators to create efficient, flexible, and highly available servers that empower organizations with the power of open source. As Ubuntu administrators, we're in good company—according to W3Techs, Ubuntu is the most widely deployed distribution on the web with regard to Linux. With the release of Ubuntu 22.04, this platform becomes even more exciting!

In this book, we will dive right into Ubuntu Server, and you will learn all the concepts needed to manage your servers and configure them to perform all kinds of neat tasks, such as serving web pages, managing virtual machines, running containers, automating configuration, sharing files with other users, and even running Ubuntu in the cloud.

We'll start our journey with the first chapter, where we'll walk through the installation of Ubuntu Server 22.04, which will serve as a foundation for the rest of the book. As we proceed through our journey, we'll look at managing users, connecting to networks, and controlling processes. Later, we'll implement important technologies, such as DHCP, DNS, Apache, MariaDB, and more. We'll even set up our own Nextcloud server along the way.

Finally, the end of the book covers various things we can do to troubleshoot issues, as well as preventing and recovering from disasters.

Who this book is for

This book is intended for readers with intermediate or advanced-beginner Linux skills, who would like to learn all about setting up servers with Ubuntu Server. This book assumes that the reader knows the basics of Linux, such as editing configuration files and running basic commands. In addition, this book assumes a basic understanding of fundamental networking concepts, such as DHCP, DNS, IP Addresses, and so on. Also, this book assumes that the reader understands the purpose of basic network hardware, such as routers, switches, and similar hardware.

What this book covers

Chapter 1, Deploying Ubuntu Server, covers the installation process for Ubuntu Server. This chapter walks you through creating bootable media and the installation process.

Chapter 2, Managing Users and Permissions, covers user management in full. Topics here will include creating and removing users, configuring password policies, and using the `sudo` command, as well as group management and switching from one user to another.

Chapter 3, Managing Software Packages, takes the reader through the process of searching for, installing, and managing packages. This will include managing APT repositories and installing packages, and even a look at Snap packages.

Chapter 4, Navigating and Essential Commands, teaches you the essential commands needed necessary for navigating through directory trees, viewing the contents of log files, and perusing log files.

Chapter 5, Managing Files and Directories, expands on the knowledge gained from the previous chapter and rounds out your toolset of essential commands by going over how to edit, copy, move, and rename files.

Chapter 6, Boosting Your Command-line Efficiency, goes over additional tips, tricks, and techniques to enhance the reader's usage of the command line. Topics here include managing output, investigating Bash history, and more.

Chapter 7, Controlling and Managing Processes, teaches the reader how to manage what is running on the server, as well as how to stop misbehaving processes. This will include having a look at `htop`, `systemd`, and managing jobs.

Chapter 8, Monitoring System Resources, goes over how to manage valuable system resources on your server, such as viewing disk and memory usage, as well as understanding load average and how it impacts your CPU.

Chapter 9, Managing Storage Volumes, takes a look at storage volumes. You'll be shown how to view disk usage, format volumes, manage the `/etc/fstab` file, use LVM, and more. In addition, we'll look at managing swap.

Chapter 10, Connecting to Networks, takes a look at networking in Ubuntu, specifically how to connect to resources from other nodes. We'll look at assigning IP addresses, connecting to other nodes via OpenSSH, and name resolution.

Chapter 11, Setting Up Network Services, revisits networking with more advanced concepts. In this chapter, the reader will learn more about the technologies that glue our network together, such as DHCP and DNS. The reader will set up their own DHCP and DNS server, as well as installing NTP.

Chapter 12, Sharing and Transferring Files, is all about sharing files with others. Concepts will include the setup of Samba and NFS network shares, and we will even go over transferring files manually with `rsync` and `scp`.

Chapter 13, Managing Databases, takes the reader through the journey of setting up and managing databases via MariaDB. The reader will learn how to install MariaDB, how to set up databases, and how to create a secondary database server.

Chapter 14, Serving Web Content, takes a look at serving content with Apache. In addition, the reader will be shown how to secure Apache with an SSL certificate, manage modules, and even set up a Nextcloud server by the end of the chapter.

Chapter 15, Automating Server Configuration with Ansible, will show the reader how to set up a Git repository for holding configuration management scripts, how to use the powerful Ansible tool to automate common administrative tasks, and also how to use `ansible-pull`.

Chapter 16, Virtualization, is all about virtualization (unsurprisingly!). The reader will be walked through setting up their very own KVM installation, as well as how to manage virtual machines with `virt-manager`.

Chapter 17, Running Containers, discusses the subject of containers and shows the reader how to manage containers in both Docker and LXD.

Chapter 18, Container Orchestration, teaches you how to take containers to the next level and manage them with the power of Kubernetes. You'll not only learn how to install Micro K8s, but also how to build your very own Kubernetes cluster from scratch.

Chapter 19, Deploying Ubuntu in the Cloud, shows you how to spin up Ubuntu servers in the cloud, with an introduction to **Amazon Web Services (AWS)**.

Chapter 20, Automating Cloud Deployments with Terraform, goes over the process of automating the process of building cloud infrastructure using Terraform.

Chapter 21, Securing Your Server, takes a look at various things the reader can do to strengthen security on Ubuntu servers. Topics will include lowering the attack surface, securing OpenSSH, setting up a firewall, and more.

Chapter 22, Troubleshooting Ubuntu Servers, consists of topics relating to things we can do when our deployments don't go according to plan. The reader will also investigate the problem space, view system logs, and trace network issues.

Chapter 23, Preventing Disasters, informs the reader of various strategies that can be used to prevent and recover from disasters. This includes a look at utilizing Git for configuration management, implementing a backup plan, and more.

To get the most out of this book

This book is for readers who already have some experience with Linux, though it doesn't necessarily have to be with Ubuntu. Preferably, the reader will understand basic Linux command-line skills, such as changing directories, listing contents, and issuing commands as regular users or with root. It's also preferred that the reader understands foundational networking concepts, such as the purpose of things like DHCP, DNS, as well as IP addressing. Even if you feel as though you need a refresher, you should read this book anyway—the opening chapters will cover many of these concepts.

In this book, we'll take a look at real-world situations in which we can deploy Ubuntu Server. This will include the installation process, serving web pages, setting up databases, and much more. Specifically, the goal here is to be productive. Each chapter will teach the reader a new and valuable concept, using practical examples that are relative to real organizations. Basically, we focus on getting things done, rather than focusing primarily on theory. Although the theory that goes into Linux and its many distributions is certainly interesting, the goal here is to get you to the point where if a work colleague or client asks you to perform work on an Ubuntu-based server, you'll be in a good position to get the task done. Therefore, if your goal is to get up and running with Ubuntu Server and learn the concepts that really matter, this book is definitely for you.

To follow along, you'll either need a server on which to install Ubuntu Server, a virtual Ubuntu instance from a cloud provider, or a laptop or desktop capable of running at least one virtual machine.

Download the example code files

The code bundle for the book is hosted on GitHub at <https://github.com/PacktPublishing/Mastering-Ubuntu-Server-Fourth-Edition>

We also have other code bundles from our rich catalog of books and videos available at <https://github.com/PacktPublishing/>. Check them out!

Download the color images

We also provide a PDF file that has color images of the screenshots/diagrams used in this book. You can download it here: https://static.packt-cdn.com/downloads/9781803234243_ColorImages.pdf

Conventions used

There are a number of text conventions used throughout this book.

CodeInText: Indicates code words in text, database table names, folder names, filenames, file extensions, pathnames, dummy URLs, user input, and Twitter handles. Here is an example: “Anyway, back to our `/etc/passwd` file. The fifth column is designated for user info, most commonly the user's first and last names.”

A block of code is set as follows:

```
description: External access profile
devices:
  eth0:
```

```
name: eth0
nictype: bridged
parent: br0
type: nic
```

When we wish to draw your attention to a particular part of a code block, the relevant lines or items are highlighted:

```
- name: Start the apache2 services
  ansible.builtin.service:
    name: apache2
    state: started
    enabled: true
```

Any command-line input or output is written as follows:

```
sudo apt install docker.io
```

Bold: Indicates a new term, an important word, or words that you see on the screen. For instance, words in menus or dialog boxes appear in the text like this. For example: “Select **System info** from the **Administration** panel.”



Warnings or important notes appear like this.



Tips and tricks appear like this.

Get in touch

Feedback from our readers is always welcome.

General feedback: Email feedback@packtpub.com and mention the book's title in the subject of your message. If you have questions about any aspect of this book, please email us at questions@packtpub.com.

Errata: Although we have taken every care to ensure the accuracy of our content, mistakes do happen. If you have found a mistake in this book, we would be grateful if you reported this to us. Please visit <http://www.packtpub.com/submit-errata>, click **Submit Errata**, and fill in the form.

Piracy: If you come across any illegal copies of our works in any form on the internet, we would be grateful if you would provide us with the location address or website name. Please contact us at copyright@packtpub.com with a link to the material.

If you are interested in becoming an author: If there is a topic that you have expertise in and you are interested in either writing or contributing to a book, please visit <http://authors.packtpub.com>.

Share your thoughts

Once you've read *Mastering Ubuntu Server, Fourth Edition*, we'd love to hear your thoughts! Please [click here](#) to go straight to the Amazon review page for this book and share your feedback.

Your review is important to us and the tech community and will help us make sure we're delivering excellent quality content.

1

Deploying Ubuntu Server

Ubuntu Server is an extremely powerful distribution of Linux for servers and network appliances. Whether you're setting up a high-end database or a small office file server, the flexible nature of Ubuntu Server will meet and surpass your needs. In this book, we'll walk through all the common use cases to help you get the most out of this exciting platform. Ubuntu Server features a perfect mix of modern development frameworks and rock-solid stability, and its hardware support enables it to be installed on the latest server hardware.

In this chapter, I'll guide you through the process of deploying Ubuntu Server from start to finish. We'll begin with some discussion of best practices, and then we'll obtain the software and create our installation media. Next, I'll give you a step-by-step rundown of the entire installation procedure. By the end of this chapter, you'll have an Ubuntu Server installation of your own to use throughout the remainder of this book. In addition, since Canonical (the makers of Ubuntu) now offers official support for Raspberry Pi, we'll look at the process of setting that up as well.

In this chapter, we will cover:

- Technical requirements
- Determining your server's role
- Choosing a device for our server
- Obtaining installation media
- Creating a bootable flash drive
- Installing Ubuntu Server
- Installing Ubuntu Server on a Raspberry Pi

To get started, we'll first take a look at some of the technical requirements for deploying a server with Ubuntu.

Technical requirements

To follow along with the examples in this book, you'll need an Ubuntu Server installation to work with. In general, the following specifications are the estimated minimums to successfully install Ubuntu Server:

- 64-bit CPU
- 1 GB RAM
- 10-GB hard disk (16 GB or more is recommended)

64-bit CPU support is now a requirement, with the only exception being the Raspberry Pi version. This is because Canonical no longer makes versions of Ubuntu available for 32-bit PC and server processors. While this may seem like a downside, all computers sold today support 64-bit operating systems, and consumer CPUs have been 64-bit capable since at least 2003. Even if you have an older PC lying around that you don't think is capable of running a 64-bit operating system, you'd be surprised—even the later models of the Pentium IV (which is quite old) support this, so this requirement shouldn't be hard to meet. Don't worry about the particulars of this right now, we'll go through the requirements in more detail later on in this chapter.

Now that we understand the technical requirements of Ubuntu Server, let's consider the role our server will play in our organization.

Determining your server's role

You're excited to set up an Ubuntu Server installation so you can follow along with the examples contained in this book. It is also important to understand how a typical server rollout is performed in the real world. Every server must have a purpose, or *role*. This role could be that of a database server, web server, file server, and so on. In a nutshell, the role is the value the server adds to you or your organization. Sometimes, servers may be implemented solely for the purpose of testing experimental code. And this is important too—having a test environment is a very common (and worthwhile) practice.

Once you understand the role your server plays within your organization, you can plan for its implementation. Is the system mission critical? How would it affect your organization if for some reason this server malfunctioned? Depending on the answer to this question, you may only need to set up a single server for this task, or you may wish to plan for redundancy such that the server doesn't become a central point of failure. An example of this may be a DNS server, which would affect your colleagues' ability to resolve local hostnames and access required resources. It may make sense to add a second DNS server to take over in the event that the primary server becomes unavailable for some reason.

Another item to consider is how confidential the data residing on a server is going to be for your environment. This directly relates to the installation procedure we're about to perform, because you will be asked whether or not you'd like to utilize encryption. The encryption that Ubuntu Server offers during installation is known as encryption at rest, which refers to the data stored within the internal storage volumes on that server. If your server is destined to store confidential data (accounting information, credit card numbers, employee or client records, and so on), you may want to consider making use of this option.

Encrypting your hard disks is a really good idea to prevent miscreants with local access from stealing data. As long as the attacker doesn't have your encryption key, they cannot steal this confidential information. However, it's worth mentioning that anyone with physical access can easily destroy data (encrypted or not), so remember to keep your server room locked!

At this point in the book, I'm definitely not asking you to create a detailed implementation diagram or anything like that, but instead to keep in mind some concepts that should always be part of the conversation when setting up a new server. It needs to have a reason to exist, it should be understood how critical and confidential the server's data will be, and the server should then be set up accordingly. Once you practice these concepts as well as the installation procedure, you can make up your own server roll-out plan to use within your organization going forward. All in all, understanding the purpose of each component in your infrastructure is a great mindset to adopt.

At this point, we now understand how we might identify a role for our server and how it will fit into our organization. In the next section, we'll take a look at the process of actually installing Ubuntu Server, so we will have at least one test machine to use for the examples in this book.

Choosing a device for our server

I bet you're excited to set up your very own installation of Ubuntu Server and dive in. But before we can do that, we have to decide what to actually install it on. For the purposes of this book, there isn't a specific requirement in terms of hardware. You just need an Ubuntu Server installation of some sort, and it wouldn't hurt to set up multiple servers if you can—you don't need them all to be on the same device type. Having multiple servers will help you experiment with networking when we get to that point later on in the book. But for now, it's only a matter of utilizing whatever you have at your disposal to get an Ubuntu installation going.

In particular, the following list includes the most common devices you can consider for your Ubuntu Server installation:

- Physical server
- Physical desktop
- Laptop
- Virtual machine
- Virtual private server
- Raspberry Pi

Let's take a look at each of these options in more detail.

Physical server

Nowadays, it's very easy to find used physical servers for an affordable price. Dell PowerEdge is a very common model, and the R610 and R710 specifically are good choices that are readily available second. These servers are commonly made available in the reseller market after companies upgrade to newer models. The R610 and R710 are a bit old, but their specs are still great for testing purposes. If you're able to find a newer model (such as the R720) for a reasonable price, even better.

The downside with physical servers is that they take up a lot of room and can often be power-hungry (and noisy). Make sure to shut them down when not in use and look into the cost of electric services in your area—these servers can be very cheap to run or very expensive, depending on your electricity rates.

Physical desktop

If you don't have access to a physical server, you might consider running Ubuntu Server on a desktop. It's common for some computer users to hang onto their older PC after upgrading to something new and shiny. So rather than let your old desktop collect dust, why not put it to work? Sure, your older computer may not run today's high-end games, but that doesn't matter for our purposes. Ubuntu Server runs very well, even on older hardware. In fact, it's quite common for home learners to use small form factor PCs, such as the Intel NUC, for this purpose. In addition, using a physical desktop has some advantages over physical servers as well. They often use much less power than server hardware, and they also tend to produce less noise.

When it comes to production servers (installations of Ubuntu Server for use within your organization), desktops are typically not a good pick. Depending on the model, actual server hardware might have additional hardware and features that might not be present with a desktop. For example, true server builds often feature true hardware RAID, **Error Correction Code (ECC)** memory, multiple processors, and more. Although a desktop tower typically lacks those features, if you install Ubuntu Server on your old PC it's every bit as valid a server as an actual server chassis would be – though it wouldn't scale well in a data center. Since we're just learning though, that doesn't matter for us for our current use-case.

Laptop

Another option worth considering is to install Ubuntu Server on a laptop. If you have an older laptop lying around that you're no longer using, it might be a great option for learning Ubuntu Server. If you do decide to use a laptop for this purpose, then the same factors will apply to this as I wrote regarding using a desktop.

However, the reason I decided to give laptops their own section here is because there are some additional benefits with these that you can take advantage of. With a real data center server rack, you'll typically have access to a **Keyboard, Video, and Mouse (KVM)**. This might mean that you could have a physical monitor, keyboard, and mouse attached to the server, or a special device with all three built in. Servers within data centers will also have an **Uninterruptible Power Supply (UPS)** that will keep them running for a time, even when your power goes out.

When it comes to a laptop, it has those same features. Laptops have a built-in keyboard, mouse, and display. If the battery that's installed in the laptop works, then you have a UPS as well. For those reasons, a laptop might even have an advantage over desktops overall.

However, just like desktop computers, laptops aren't generally acceptable for use in an actual data center. For our use-case though, we only need one or more installations of Ubuntu Server for going through the examples in this book. And for that purpose, dedicating any computers you're not currently using would be more than adequate.

Virtual machine

If you don't have access to a physical machine, you might consider a **Virtual Machine (VM)**. Most computers sold nowadays support the ability to run VMs. **VirtualBox** is a great solution, as it's easy to use and available for all of the major operating systems. Just like Ubuntu itself, VirtualBox is available for free, so it's typically the lowest-cost entry-point for getting started. Also, VirtualBox allows you to easily create snapshots of your Ubuntu installation, so you can create a point-in-time backup before going through an example in this book, and then restore it to repeat tasks as often as you'd like. The ability to utilize snapshots alone makes a VM especially attractive for our needs.

The downside to VirtualBox is that you'll need to be able to dedicate at least 1 GB of RAM to your Ubuntu Server VM, and your CPU will need to support virtualization extensions, which you'll need to enable in your computer's settings if your device supports it.

VirtualBox can be downloaded here: <https://www.virtualbox.org>.

Virtual private server

Services such as Amazon Web Services, Google Cloud, Linode, Microsoft Azure, Digital Ocean, and others allow you to set up Ubuntu Server in the cloud for you to connect to and manage via OpenSSH. Choosing the **Virtual Private Server (VPS)** option has some benefits; you don't need to find room for a large physical server, and you don't need to worry about power usage either. Another benefit is that you won't even need to go through the installation process in this chapter at all; the cloud provider will do that for you when you choose to deploy Ubuntu Server on their platform. The main downside, though, is that VPS instances are not free—you'll need to look at the costs associated with running Ubuntu on such a server and decide if that cost makes sense. Some VPS services allow you to set up instances for as little as \$5 USD per month, which can be lower than the cost of the electricity needed to run a physical server in some areas.

Raspberry Pi

Raspberry Pi units are quickly becoming a user favorite for server use cases. They are inexpensive (some models are available for less than \$40 USD) and they also use very minimal electricity—you can leave them powered on 24/7 with a virtually unnoticeable difference in your power bill. In fact, they use about as much power as it takes to charge a high-end cellphone. Another benefit is that a Raspberry Pi is generally going to be more powerful than an entry-level VPS. The cheapest VPS instances typically have just 1 CPU core and 1 GB of RAM, but modern Raspberry Pis are produced with quad-core CPUs, as well as 2 GB, 4 GB, or 8 GB of RAM depending on which model you purchase. This means that a Raspberry Pi can possibly perform better than cheaper VPS instances. The downside to Raspberry Pi is that some applications are unavailable, since they utilize ARM CPUs instead of x86. This means that some examples in this book will not work on a Pi (although the majority will).

Once you've chosen a device to install Ubuntu Server on, we can continue. If you chose to utilize a VPS, you can move on to the next chapter, *Chapter 2, Managing Users and Permissions*, as you won't need to walk through the installation process. In the case of the Raspberry Pi, if that's your chosen platform, you can skip to the end of this chapter for a dedicated section about setting that up. For all other devices, continue reading for a walk-through for the Ubuntu Server Live Installer.

Obtaining installation media

It's time to get started! If you've decided to utilize a physical server, desktop, laptop, or VM as your test server, then you'll need to go through the installation process to set up Ubuntu. Don't worry—it's very easy to do and it might even be easier than you'd think, as there are fewer overall steps in the process when compared to older versions. If you've instead opted to use a VPS or Raspberry Pi, you won't need to go through this process, as VPS providers do this for you and Raspberry Pi has a different setup method altogether (we'll cover this later in the chapter, in the *Installing Ubuntu on a Raspberry Pi* section).

Assuming that you've decided to use a device that does require going through the installer, we'll need to download Ubuntu Server and then create bootable installation media to install it. How you do this largely depends on your hardware. Does your device have an optical drive? Is it able to boot from USB? Refer to the documentation for your device to find out.

It's recommended to utilize a flash drive for the installation if you can, preferably one that uses USB 3.0 or higher since you'd benefit from its faster speed compared to USB 2.0. The reason for the preference for using a flash drive is due to the fact that they are typically faster than a DVD.

However, if your device is older, you won't have a choice, as legacy devices are not able to boot from USB at all. As a general rule, use a flash drive if you can and opt for a DVD only if you have no choice.

In the past, Ubuntu Server ISO images could be used to create either a bootable CD or DVD. Nowadays, writable CDs don't have enough space to support the download size. Therefore, if you choose to burn to bootable optical media, you'll need a writable DVD at a minimum.

Unfortunately, the differing age of servers within a typical data center introduces some unpredictability when it comes to how to boot installation media. When I first started with servers, it was commonplace for all standard rack servers to contain a 3.5-inch floppy disk drive, and some of the better ones even contained an optical drive. Nowadays, servers typically contain neither. If a server does have an optical drive, it could potentially go unused for an extended period of time and become faulty without anyone knowing until the next time someone goes to use it. Some servers boot from USB, others don't. To continue, check the documentation for your hardware and plan accordingly. Your server's capabilities will determine which kind of media you'll need to create.

Regardless of whether we plan on creating a bootable USB or DVD, we only need to download a single file. Navigate to the following site in your web browser to get started: <https://www.ubuntu.com/download/server>.

From this page, we're going to download Ubuntu 22.04 LTS by clicking on the **Option 2 – Manual server installation** button:

Get Ubuntu Server

Option 1: Instant Ubuntu VMs

Try Multipass, a mini cloud on Mac, Windows and Linux

- Always up-to-date with security fixes
- Cloud-init metadata for cloud dev and test
- Virtualbox, Hyper-V, HyperKit or KVM



[Get Multipass](#)

[Option 1 - Instant Ubuntu VMs](#) [Option 2 - Manual server installation](#) [Option 3 - Automated server provisioning](#)

Figure 1.1: Ubuntu Server 22.04 download page


Next, you'll be offered one or more versions of Ubuntu Server for download:

Get Ubuntu Server

Option 2: Manual server installation

USB or DVD image based physical install

- OS security guaranteed until April 2025
- Extended security maintenance until April 2030
- Commercial support for enterprise customers



[Download Ubuntu Server 22.04 LTS](#) [Alternative downloads](#) [Alternative architectures](#)

[Read the Ubuntu Server 22.04 LTS release notes](#)

[Option 1 - Instant Ubuntu VMs](#) [Option 2 - Manual server installation](#) [Option 3 - Automated server provisioning](#)

Figure 1.2: Revealing Ubuntu Server 22.04 download options

As of the time this chapter was being written, Ubuntu Server 22.04 is the latest version, and is shown as the primary download option. There may be other versions of Ubuntu Server listed on this page, such as 22.10 and 23.04, depending on when you're reading this. New releases of Ubuntu are published every six months. However, this book only covers the **Long Term Support (LTS)** release, due to the fact that it benefits from five years of support.

By comparison, non-LTS versions (or “interim releases”) are only supported for nine months and are therefore not an appropriate choice for production servers. Organizations don't typically utilize non-LTS releases at all, except for testing upcoming features prior to general availability, so for our purposes, we'll stick with the LTS version. Once the download is completed, we'll end up with an ISO image we can use to create our bootable installation media.

If you're setting up a VM, then the ISO file you download from the Ubuntu downloads page will be all you need; you won't need to create a bootable DVD or flash drive. In that scenario, all you should need to do is create a VM, attach the ISO to the virtual optical drive, and boot it. From there, the installer should start, and you can proceed with the installation procedure outlined later in this chapter, in the *Installing Ubuntu Server* section. Going over the process of booting an ISO image on a VM differs from one virtualization solution to another, so detailing the process with each would be a very difficult task. Thankfully, the process is usually straightforward and you can find the details within the documentation of your hypervisor. In most cases, the process is as simple as attaching the downloaded ISO image to the VM and then starting it up.

If your device does not support booting from USB and you find yourself needing to create a bootable DVD, the process is typically just a matter of downloading the ISO file and then right-clicking on it. In the right-click menu of your operating system, you should have an option to burn to disk or some similar verbiage. This is true of Windows, as well as most graphical desktop environments of Linux where a disk-burning application is installed.

The exact procedure differs from system to system, mainly because there is a vast amount of software combinations at play here. For example, I've seen many Windows systems where the right-click option to burn a DVD was removed by an installed CD/DVD-burning application. In that case, you'd have to first open your CD/DVD-burning application and find the option to create media from a downloaded ISO file. As much as I would love to outline the complete process here, no two Windows PCs typically ship with the same CD/DVD-burning application. The best rule of thumb is to try right-clicking on the file to see whether the option is there, and, if not, refer to the documentation for your application. Keep in mind that a *data disk* is not what you want, so make sure to look for the option to create media from an ISO image or your disk will be useless for our purpose.

At this point, you should have an Ubuntu Server ISO image file downloaded. If you are planning on using a DVD to install Ubuntu, you should have that created as well. In the next section, I'll outline the process of creating a bootable flash drive that can be used to install Ubuntu Server.

Creating a bootable flash drive

The process of creating a bootable USB flash drive with which to install Ubuntu used to vary greatly between platforms. The steps were very different depending on whether your workstation or laptop was currently running Linux, Windows, or macOS. Thankfully, a much simpler method has come about. Nowadays, I recommend the use of **Etcher** to create your bootable media. Etcher is fantastic in that it abstracts the method such that it is the same regardless of which operating system you use, and it distills the process to its most simple form.

Another feature I like is that Etcher is safe; it prevents you from destroying your current operating system in the process of mastering your bootable media. In the past, you'd use tools like the `dd` command on Linux to write an ISO file to a flash drive. However, if you set up the `dd` command incorrectly, you could effectively write the ISO file over your current operating system and wipe your entire hard drive. Etcher doesn't let you do that.

Before continuing, you'll need a USB flash drive that is either empty or one you don't mind wiping. This process will completely erase its contents, so make sure the device doesn't have information on it that you might need. The flash drive should be at least 2 GB or larger. Considering it's difficult to find a flash drive for sale with less than 4 GB of space nowadays, this should be relatively easy to obtain.

To get started, head on over to <https://www.balena.io/etcher/>, download the latest version of the application from their site, and open it up. The window will look similar to the following screenshot once it launches:

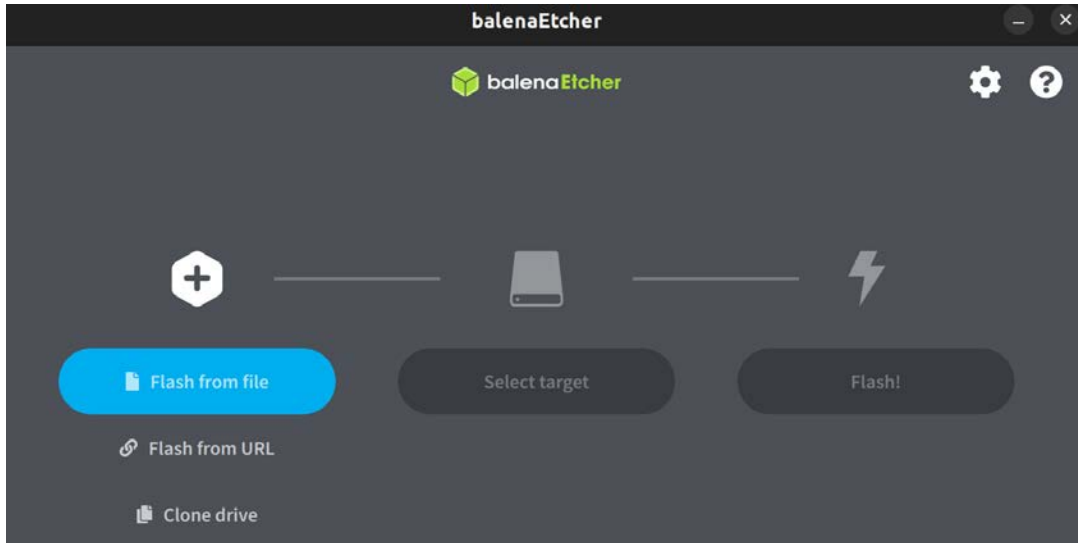


Figure 1.3: Utilizing Etcher to create a bootable flash drive

At this point, click **Flash from file**, which will open up a new window that will allow you to select the ISO file you downloaded earlier. Once you select the ISO, click on **Open**:

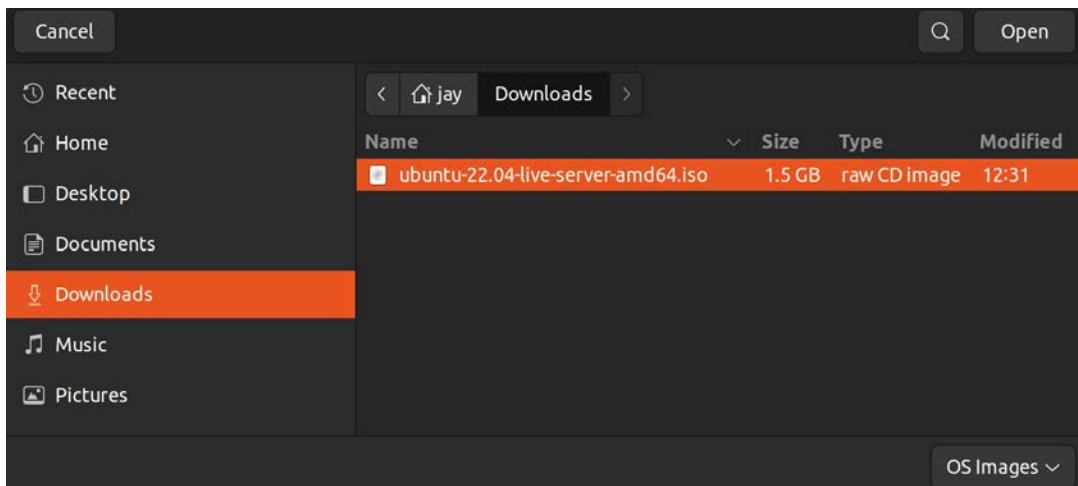


Figure 1.4: Selecting an ISO image with Etcher

If your flash drive is already inserted into the computer, Etcher should automatically detect it. In the event that you have more than one flash drive attached, or Etcher selects the wrong one, you can click **Change** and select the flash drive you wish to use:

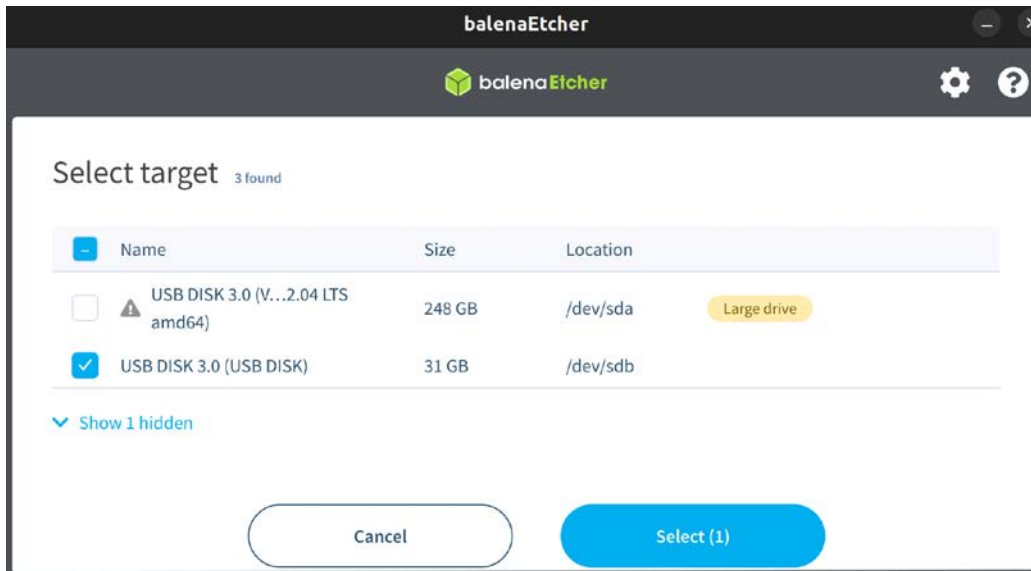


Figure 1.5: Selecting a different flash drive with Etcher

Finally, click **Continue** to get the process started. At this point, the flash drive will be converted into Ubuntu Server installation media that can then be used to start the installation process:

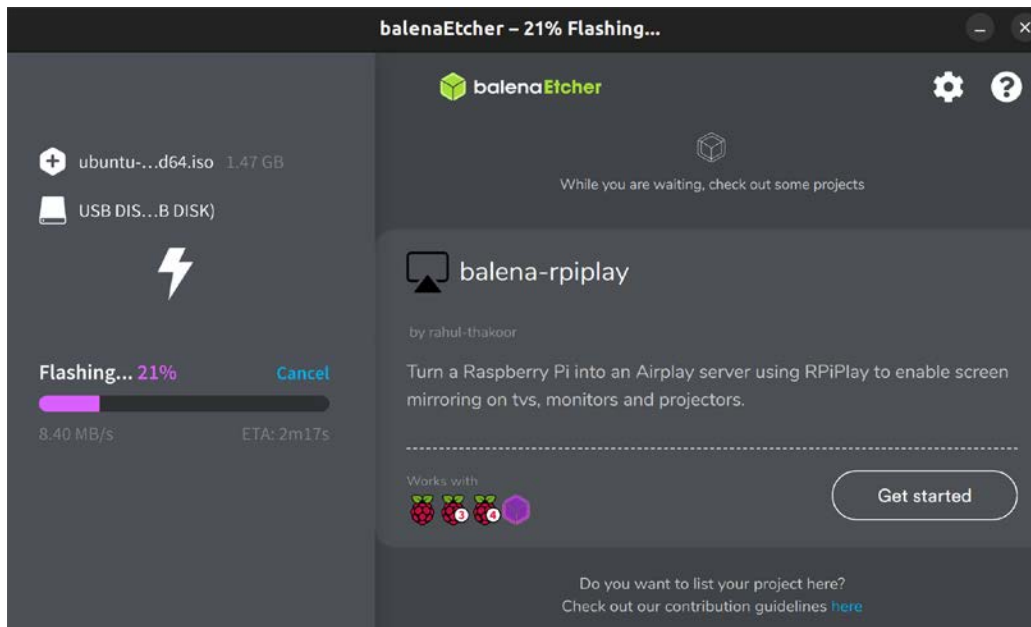


Figure 1.6: Etcher in the process of writing a flash drive

After a few minutes (the length of time varies depending on your hardware), the **Flashing** process will complete, and you'll be able to continue and get some installations going. Before we get into that, though, we should have a quick discussion regarding partitioning.

Planning the partitioning layout

Partitioning your disk allows you to divide up your hard disk to dedicate specific storage allocations to individual applications or purposes. For example, you can dedicate a partition for the files that are shared by your Apache web server so changes to other partitions won't affect it. You can even dedicate a partition for your network file shares—the possibilities are endless. Each partition is mounted (attached) to a specific directory, and any files sent to that directory are thereby sent to that separate partition. The names you create for the directories where your partitions are mounted are arbitrary; it really doesn't matter what you call them. The flexible nature of storage on Linux systems allows you to be creative with your partitioning as well as your directory naming, since the Linux filesystem gives you more flexibility when it comes to mounting storage devices in specific folders.

Admittedly, we're getting ahead of ourselves here. After all, we're only just getting started and the point of this chapter is to help you set up a basic Ubuntu Server installation to serve as the foundation for the rest of the chapter. When going through the installation process, we'll accept the defaults anyway. However, the goal of this section is to give you examples of the options you have for consideration later. At some point, you may want to get creative and play around with the partition layout.

With custom partitioning, you're able to do some very clever things. For example, with the right configuration, you'll be able to wipe and reload your distribution while preserving user data, logs, and more. This works because Ubuntu Server allows you to carve up your storage any way you want during installation. If you already have a partition with data on it, you can choose to leave it as is so you can carry it forward into a new install. You simply set the directory path where it's mounted to be the same as before, restore your configuration files, and your applications will continue working as if nothing happened.

One very common example of custom partitioning in the real world is separating the `/home` directory into its own partition. Since this is where users typically store their files, you can set up your server such that a reload of the distribution won't disturb their files. When they log in after a server refresh, all their files will be right where they left them. You could even place the files shared by your Apache web server on to their own partition and preserve those too. You can get very creative here.

It probably goes without saying, but when reinstalling Ubuntu, you should back up partitions that have data you don't want to be wiped (even if you don't plan on formatting the partitions). The reason being, one wrong move (literally a single checkbox) and you can easily wipe out all the data on that partition. Always back up your data when refreshing a server.

Another reason to utilize separate partitions may be to simply create boundaries or restrictions. If you have an application running on your server that is prone to filling up large amounts of storage, you can point that application to its own partition, limited by size. An example of where this could be useful is an application's log files. Log files are the bane of any system administrator's life when it comes to storage.

While helpful if you're trying to figure out why something crashed, logs can fill up a hard disk very quickly if you're not careful. In my experience, servers have been known to come to a screeching halt due to log files filling up all the available free space on a server where everything was on a single partition. The only boundary the application had was the entirety of the disk itself.

While there are certainly better ways of handling excessive logging (log rotating, disk quotas, and so on), a separate partition would certainly help. If the application's log directory was on its own partition, it would be able to fill up that partition, but not the entire drive, which would cause issues but wouldn't affect the entire server. As an administrator, it's up to you to weigh the pros and the cons of these strategies to avoid server overload and develop a partitioning scheme that will best serve the needs of your organization.

Success when maintaining a server is a matter of efficiently managing resources, users, and security—a good partitioning scheme is certainly part of that. Sometimes it's just a matter of making things easier on yourself so that you have less work to do should you need to reload your operating system. For the sake of following along with this book, it really doesn't matter how you install or partition Ubuntu Server. The trick is simply to get it installed—you can always practice partitioning later. After all, part of learning is setting up a platform, observing any issues that may occur, and then fixing it up.

Here are some basic tips regarding partitioning:

- At a minimum, a partition for the root filesystem (designated by a forward slash) is required.
- The `/var` directory contains most log files and is therefore a good candidate for separation for the reasons mentioned previously in this section.
- The `/home` directory stores all user files. Separating this into a separate partition can be beneficial as it gives you the possibility of having your user files survive a reinstall of Ubuntu.
- If you've used Linux before, you may be familiar with the concept of a **swap partition**, which is a special partition that can act as RAM when your memory becomes full. This is no longer necessary—a **swap file** will be created automatically in newer Ubuntu releases.

When we perform our installation in the next section, we'll choose the defaults for the partitioning scheme to get you started quickly. However, I recommend you come back to the installation process at some point in the future and experiment with it. You may come up with some clever ways to split up your storage. However, you don't have to—having everything in one partition is fine too, depending on your needs.

Now that we have an understanding of what partitioning is, we should have all of the essential topics covered to enable us to actually get the installation of Ubuntu Server going. Let's go ahead and work on that now.

Installing Ubuntu Server

At this point, we should be ready to get an installation of Ubuntu Server going. In the steps that follow, I'll walk you through the process.

To get started, all you should need to do is insert the installation media into your server or device and follow the onscreen instructions to open the boot menu. The key you press at the beginning of the POST process differs from one machine to another, but it's quite often *F10*, *F11*, or *F12*. Refer to your documentation if you are unsure, though most computers and servers tell you which key to press at the beginning. You may miss this window of opportunity the first few times, and that's fine—to this day I still seem to need to restart the machine once or twice to hit the key in time.

Once you successfully boot your device while using your Ubuntu Server installation media, navigating the installer is relatively straightforward. You simply use the arrow keys to move up and down to select different options, and press *Enter* to confirm choices. The *Esc* key will allow you to exit from a sub-menu. Moving around within the installer is pretty easy once you get the hang of it.

Once the installer starts, you'll see the first of many selection screens. The first option, **Ubuntu Server**, allows you to start the Ubuntu Server installer. The second option, **Test memory**, runs a special program that helps you determine if you have any physical defects with the RAM modules installed inside your device. I always recommend testing the memory of your device once per year at least, and especially before you install an operating system for the first time. Memory issues can be rare, but you'd be surprised. It might be a good idea to test your memory just to be safe.

If you wish to test the memory of your device, you can go ahead and do so. In order to move on though, we'll need to choose the **Ubuntu Server** option in order to start the installer:

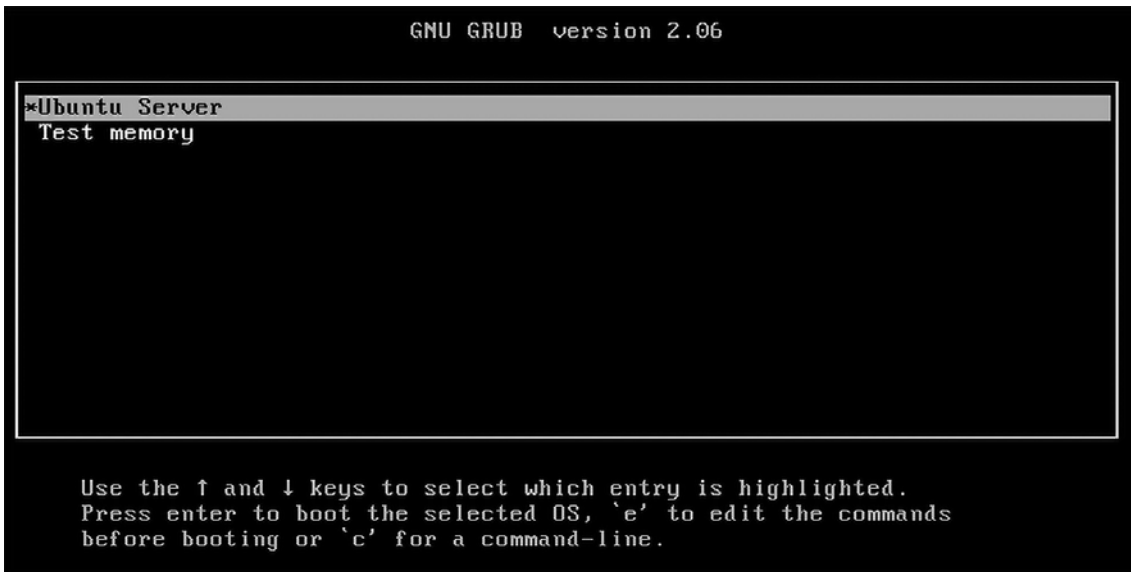


Figure 1.7: The initial boot screen for the Ubuntu Server installation media

Next, you'll see a screen allowing you to select your language as in the following screenshot:

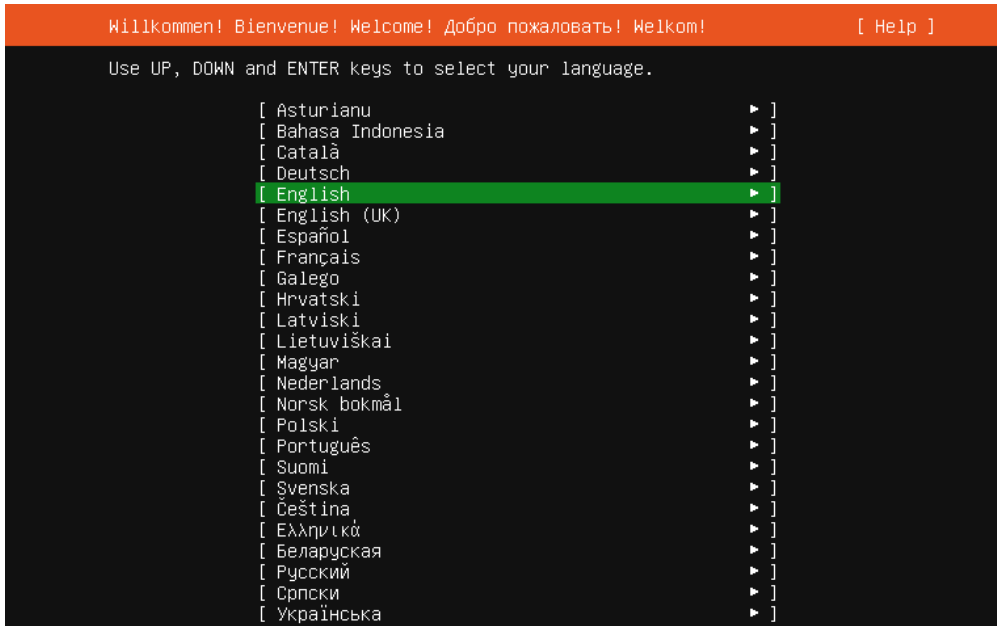


Figure 1.8: Language selection at the beginning of the installation process

If your language is anything other than English, you'll be able to select that here. Once you're satisfied with the chosen keyboard layout, you continue by selecting **Done** at the bottom of the screen.

After choosing your language, you'll be brought to a screen that enables you to set your **Keyboard configuration**. If the proper keyboard layout wasn't automatically selected, you can change it here.

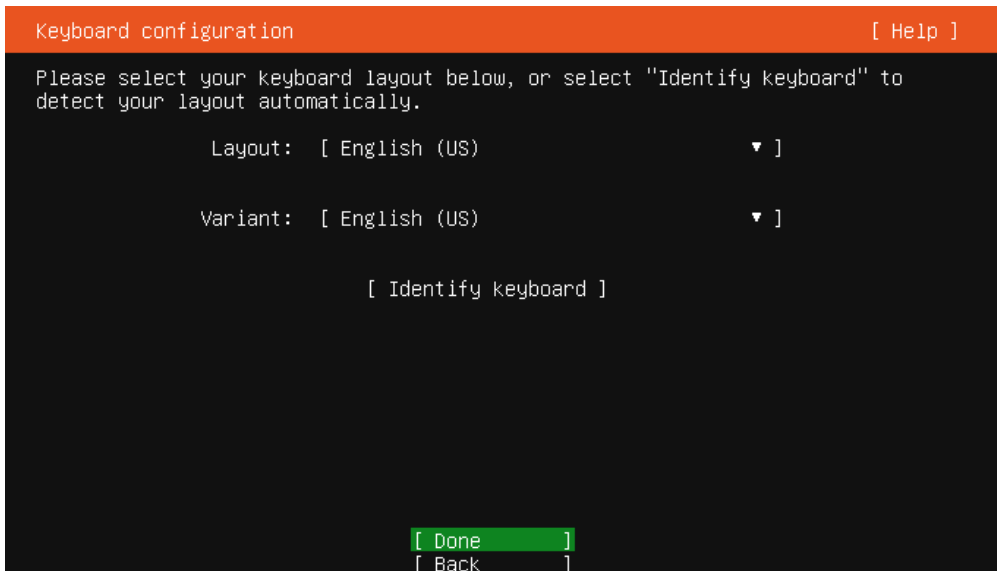


Figure 1.9: Setting the layout of your keyboard

Next, we'll see an option to install **Ubuntu Server**, or **Ubuntu Server (minimized)**. For this, we'll choose the first option. Choosing to install a minimized version of Ubuntu Server might be a good fit for creating smaller installations, however for this book, we're going to focus on the normal installation type.

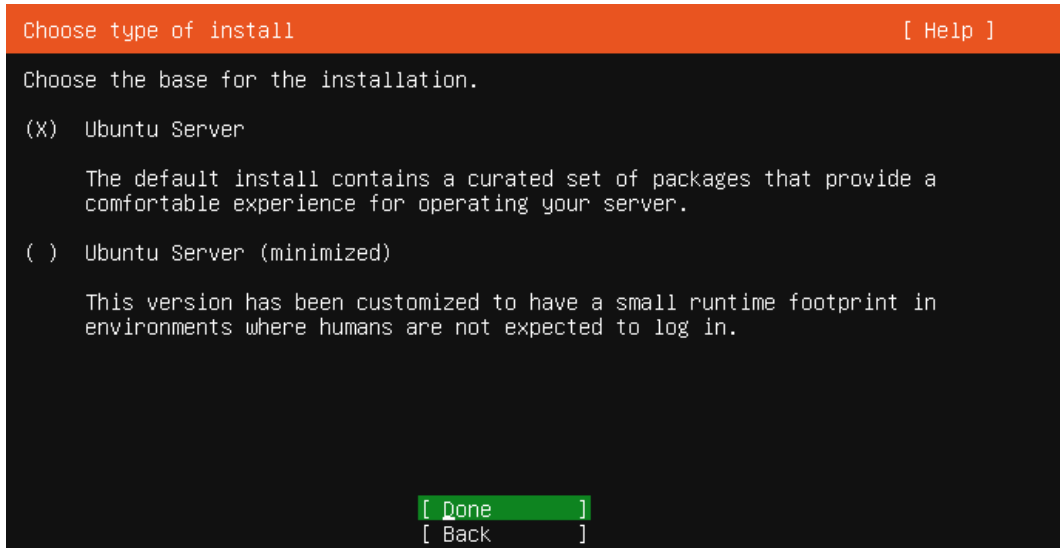


Figure 1.10: Choosing your installation type

Next, the Ubuntu Server installer will attempt to automatically detect the appropriate parameters for your network card. By default, it should detect the appropriate settings via **Dynamic Host Control Protocol (DHCP)**. We'll go over DHCP in a future chapter, specifically *Chapter 11, Setting Up Network Services*. For now, the defaults should be fine. Select **Done** to continue.

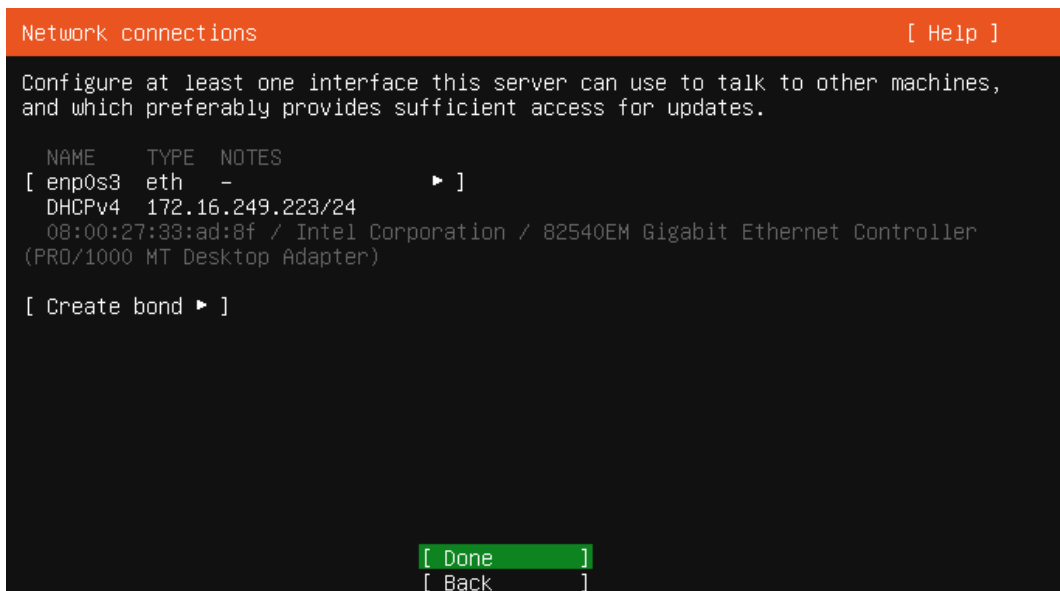


Figure 1.11: Network configuration within the Ubuntu Server installer

Continuing, the next screen will give us an opportunity to set a proxy address if we need one. Most readers shouldn't need this, and those that do might've been provided particular proxy settings from a network administrator. So for us, we'll select **Done** to continue:

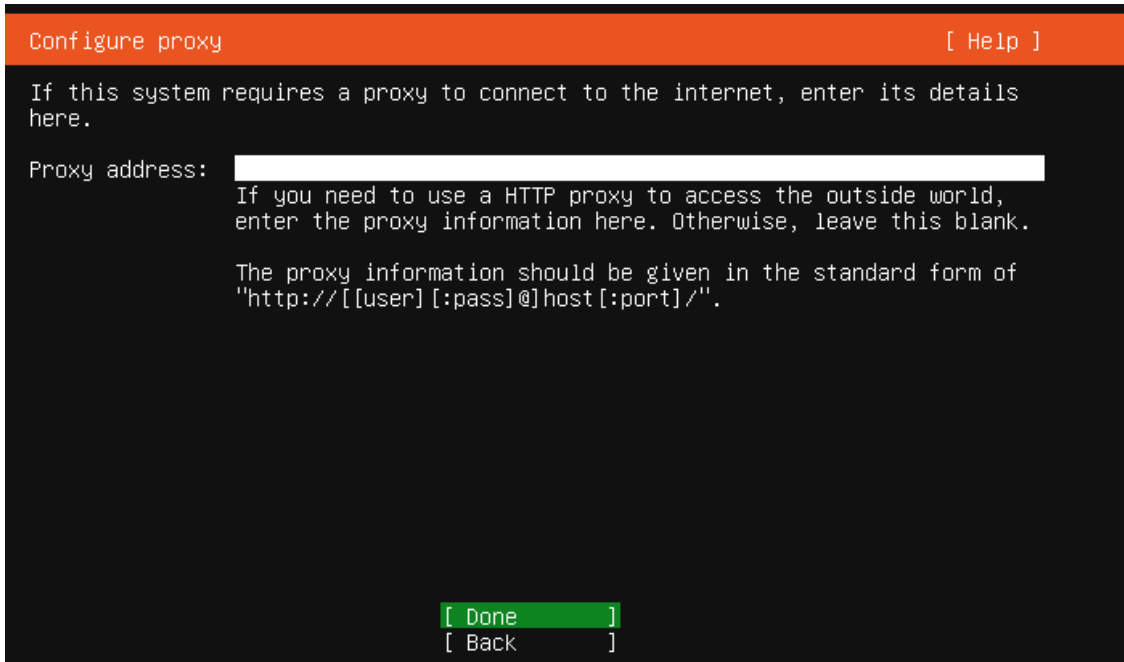


Figure 1.12: Proxy settings within the Ubuntu Server installer

On the next screen that appears, we'll see an option to set a mirror address for our installation. As we'll discuss further in *Chapter 3, Managing Software Packages*, this pertains to an online source from which installable applications (packages) can be downloaded. In some cases, an organization might host its own repository for available software.

Therefore, on this screen, you can choose a different server to provide software if you have your own. For our needs, we'll select **Done** to accept the default and continue:

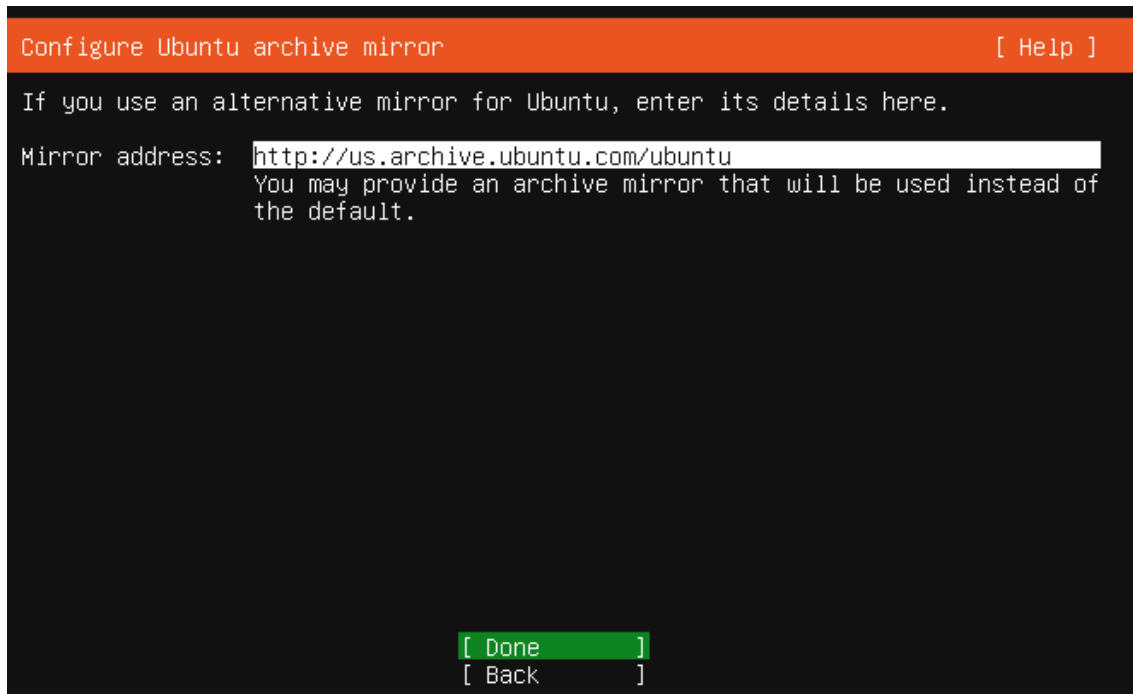


Figure 1.13: Mirror settings within the Ubuntu Server installer

Next, we'll see settings for storage. In particular, this is where we can configure the hard disk layout for our installation. Earlier in this chapter, I mentioned that it's common practice to adopt a custom partitioning scheme. If you have a preferred partition layout, this is the screen where you would input those decisions by selecting **Custom storage layout**.

However, doing so would be jumping ahead of the book a bit, so instead I recommend selecting **Done** to accept the default, which will take over the entire hard disk:

```
Guided storage configuration [ Help ]
Configure a guided storage layout, or create a custom one:
(X) Use an entire disk
    [ VBOX_HARDDISK_VB60559e99-31596435 local disk 30.000G ▼ ]
    [X] Set up this disk as an LVM group
        [ ] Encrypt the LVM group with LUKS
            Passphrase:
            Confirm passphrase:
    ( ) Custom storage layout

[ Done ]
[ Back ]
```

Figure 1.14: Configuring storage while installing Ubuntu Server

After we inform the installer how to set up the storage device on our server, we'll next see a screen that summarizes our selections while configuring storage. Since we chose the default layout, we'll see a basic layout presented on this screen. If you did decide to set up a customized partitioning scheme, you'd want to ensure that the selections shown here match what you had in mind. If you're confused as to what any of this means, don't worry too much about it for now. We'll cover more details surrounding storage in *Chapter 9, Managing Storage Volumes*. For now, select **Done** to accept the default storage layout:

```

Storage configuration [ Help ]

FILE SYSTEM SUMMARY

  MOUNT POINT      SIZE      TYPE      DEVICE TYPE
[ /                14.246G   new ext4  new LVM logical volume ▶ ]
[ /boot           1.500G   new ext4  new partition of local disk ▶ ]

AVAILABLE DEVICES

  DEVICE                                TYPE                                SIZE
[ ubuntu-vg (new)                       LVM volume group                   28.496G ▶ ]
  free space                             14.250G ▶

[ Create software RAID (md) ▶ ]
[ Create volume group (LVM) ▶ ]

USED DEVICES

  DEVICE                                TYPE                                SIZE
[ ubuntu-vg (new)                       LVM volume group                   28.496G ▶ ]
  ubuntu-lv    new, to be formatted as ext4, mounted at /  14.246G ▶

[ VBOX_HARDDISK_VB60559e99-31596435      local disk                          30.000G ▶ ]
  partition 1  new, BIOS grub spacer                            1.000M ▶
  partition 2  new, to be formatted as ext4, mounted at /boot  1.500G ▶
  partition 3  new, PV of LVM volume group ubuntu-vg            28.497G ▶

[ Done ]
[ Reset ]
[ Back ]

```

Figure 1.15: Storage configuration summary screen while installing Ubuntu Server

If you didn't already know, this process will erase everything on the hard disk of your device, and replace it with your new Ubuntu Server installation. This means that if you haven't backed up any important data that might be saved on your device, you will lose that information forever. The screen that comes up next will explain that this is the last step before your disk is wiped and Ubuntu Server is installed. If you're certain that your disk doesn't have any important data on it, you can select **Continue**:

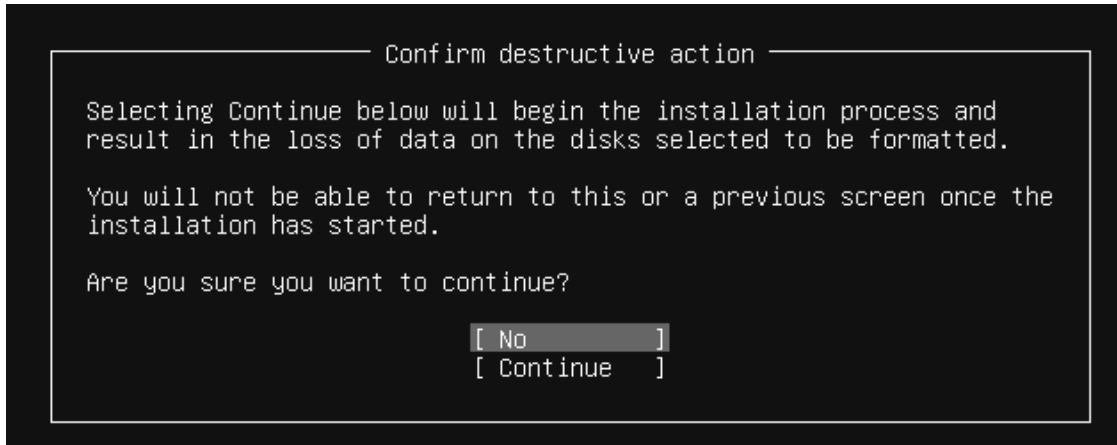


Figure 1.16: Confirming the installation and wiping of the hard disk

When we install Ubuntu Server, it will create an initial user account for us as part of the process. This user will be used for configuring and maintaining the server after installation. In this section of the installer, we'll be given an opportunity to name this user and create a password.

Fill out your information on this screen, as I have in the following screenshot. One important thing to keep in mind is that the user you create here will have administrative access. You can always create other users later on, but the user you create here will have special privileges. When finished, arrow down to **Done** and press *Enter*:

```
Profile setup [ Help ]

Enter the username and password you will use to log in to the system. You can
configure SSH access on the next screen but a password is still needed for
sudo.

Your name: Jay LaCroix

Your server's name: my-ubuntu-server
The name it uses when it talks to other computers.

Pick a username: jay

Choose a password: ****

Confirm your password: ****

[ Done ]
```

Figure 1.17: Ubuntu Server installer, configuring user details

Next, you'll see an **Install OpenSSH server** option. I recommend that you enable this. OpenSSH is the closest thing to a default we have when it comes remotely managing Ubuntu servers, and is discussed in more detail in *Chapter 10, Connecting to Networks*. For now, let's enable this option as it will be one less thing for us to do later. To do so, you can press the spacebar while the cursor is on the option to add an asterisk symbol into its checkbox, which signifies a selection. After you do that, select **Done** to continue:

```
SSH Setup [ Help ]
You can choose to install the OpenSSH server package to enable secure remote
access to your server.

  [X] Install OpenSSH server

Import SSH identity: [ No      ▼ ]
                    You can import your SSH keys from GitHub or Launchpad.

Import Username:

  [X] Allow password authentication over SSH

[ Done      ]
[ Back      ]
```

Figure 1.18: Choosing whether or not to set up OpenSSH while installing Ubuntu Server

Next, we'll see a selection of some of the more common applications that can be run on Ubuntu Server. For example, we have selections for Nextcloud, Docker, and more on this screen. For now, we'll ignore these since we'll be going through the process of manually setting up many services throughout the book. So for now, select **Done** to continue:

```

Featured Server Snaps [ Help ]

These are popular snaps in server environments. Select or deselect with SPACE,
press ENTER to see more details of the package, publisher and versions
available.

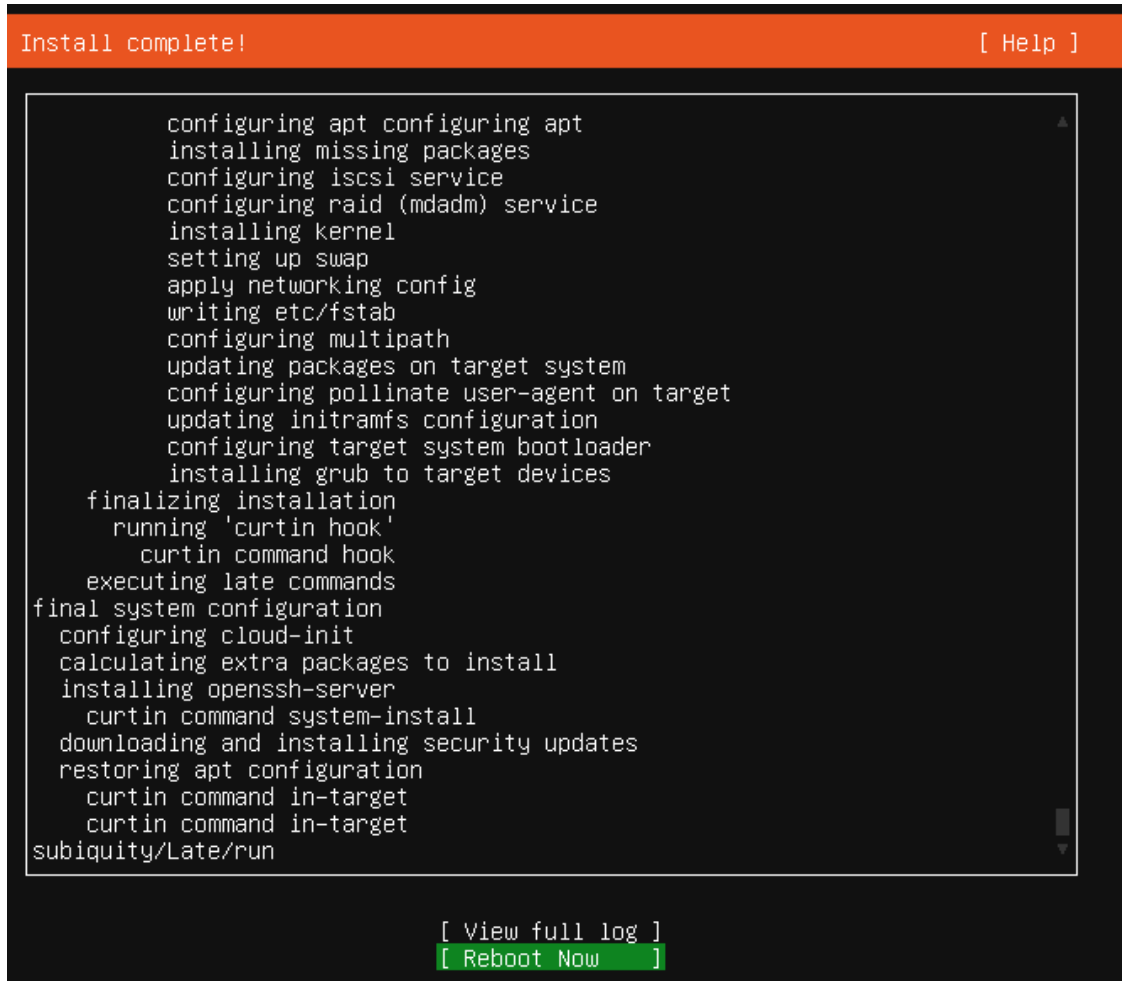
[ ] microk8s           Kubernetes for workstations and appliances
[ ] nextcloud         Nextcloud Server - A safe home for all your data
[ ] wekan             The open-source kanban
[ ] kata-containers   Build lightweight VMs that seamlessly plug into the c
[ ] docker            Docker container runtime
[ ] canonical-livepatch Canonical Livepatch Client
[ ] rocketchat-server Rocket.Chat server
[ ] mosquito         Eclipse Mosquitto MQTT broker
[ ] etcd              Resilient key-value store by CoreOS
[ ] powershell       PowerShell for every system!
[ ] stress-ng         tool to load and stress a computer
[ ] sabnzbd           SABnzbd
[ ] wormhole          get things from one computer to another, safely
[ ] aws-cli           Universal Command Line Interface for Amazon Web Servi
[ ] google-cloud-sdk  Google Cloud SDK
[ ] slcli             Python based SoftLayer API Tool.
[ ] doctl             The official DigitalOcean command line interface
[ ] conjure-up        Package runtime for conjure-up spells
[ ] postgresql10     PostgreSQL is a powerful, open source object-relatio
[ ] heroku            CLI client for Heroku
[ ] keepalived        High availability VRRP/BFD and load-balancing for Lin
[ ] prometheus        The Prometheus monitoring system and time series data
[ ] juju             Juju - a model-driven operator lifecycle manager for

[ Done ]
[ Back ]

```

Figure 1.19: Featured application selection

That's pretty much it—the installer will continue along on its own, as it has all the information it needs from you at this point. Once it's done, you'll see a **Reboot** option at the bottom of the screen; go ahead and select that:



```
Install complete! [ Help ]

configuring apt configuring apt
installing missing packages
configuring iscsi service
configuring raid (mdadm) service
installing kernel
setting up swap
apply networking config
writing etc/fstab
configuring multipath
updating packages on target system
configuring pollinate user-agent on target
updating initramfs configuration
configuring target system bootloader
installing grub to target devices
finalizing installation
  running 'curtin hook'
  curtin command hook
executing late commands
final system configuration
configuring cloud-init
calculating extra packages to install
installing openssh-server
  curtin command system-install
downloading and installing security updates
restoring apt configuration
  curtin command in-target
  curtin command in-target
subiquity/Late/run

[ View full log ]
[ Reboot Now ]
```

Figure 1.20: Completed installation, pending reboot

At this point, your server will reboot and then Ubuntu Server should start right up. Congratulations! You now have your own Ubuntu Server installation!

In addition to server-specific hardware and virtual machines, Ubuntu Server can also be installed on a Raspberry Pi, and that's exactly what we'll take a look at in the next section.

Installing Ubuntu on a Raspberry Pi

The Raspberry Pi platform has become quite a valuable asset in the industry, and a useful server platform. These tiny computers, now with four cores and up to 8 GB of RAM, are extremely power efficient and their performance is good enough that you can actually transform them into actual servers. In my lab, I have several Raspberry Pis on my network, each one responsible for performing specific tasks or functions. In many cases, you wouldn't even tell that they were devices with lower-powered hardware. Perhaps even more surprising is the fact that the Raspberry Pi 4 can outperform some lower-tier cloud instances, giving you a powerful server without the monthly cost if you don't need a high-end CPU. All this, in such a small form factor—these devices are smaller than the coaster under your coffee mug!

Ubuntu Server is available for Raspberry Pi models 2, 3, and 4. To get started, all you'll need to do is visit the official download page for the Raspberry Pi version of Ubuntu, here: <https://ubuntu.com/download/raspberry-pi>. This will take you to the following screen:

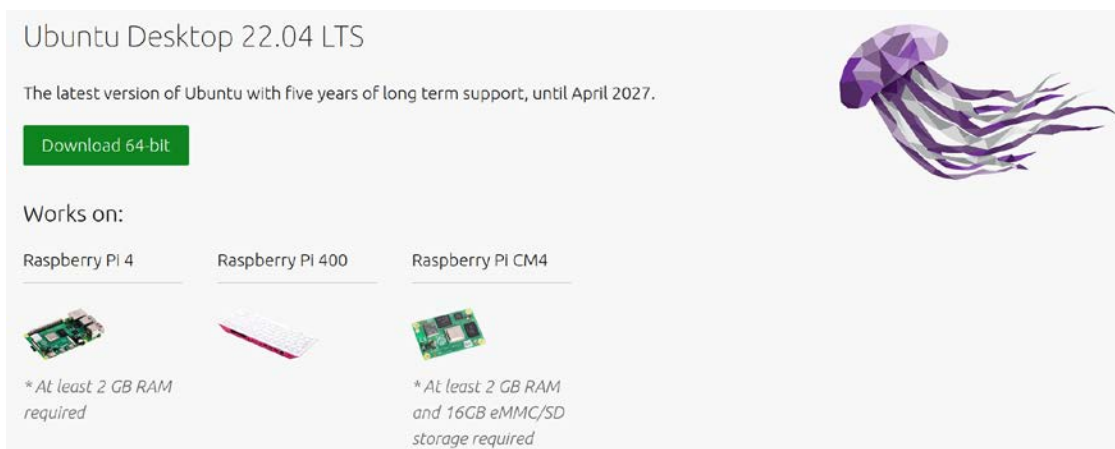


Figure 1.21: The download section for Raspberry Pi on the official Ubuntu website

Once there, you'll see a listing of multiple versions for the Pi that are available for download. Refer to the preceding screenshot for an example of options that are available, but keep in mind that Ubuntu may redesign their web page at any time. So as a rule of thumb, download the version that matches the type of Raspberry Pi you have, and go with the 64-bit version if it's available (the Raspberry Pi 2 only has a 32-bit option available).

The download for the Pi versions will come in the form of a compressed image, which can be written directly to an SD card. To get the process going, open up Etcher, which we downloaded in an earlier step. Once Etcher is open, click **Flash from file**:

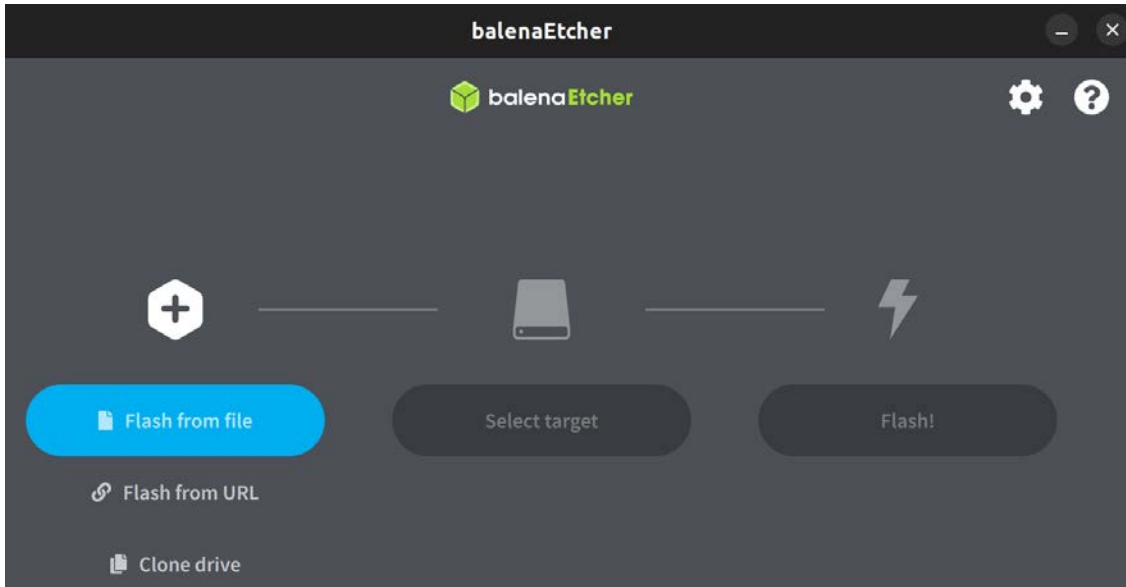


Figure 1.22: Etcher, ready for action

Now, choose the file that was downloaded earlier and select **Open**:

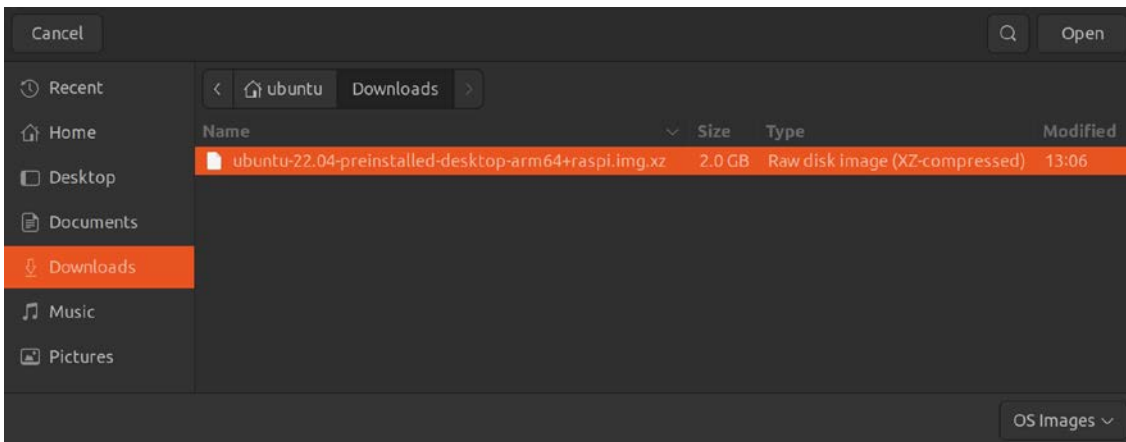


Figure 1.23: Selecting the file to use to write to the SD card

Next, go ahead and insert an SD card into your computer that will become dedicated to Ubuntu Server. Etcher will completely erase the SD card, so make sure you don't need any of the data that may be on it. If your computer does not have an SD card slot, you can use a USB card reader. If the proper device isn't selected, you can click **Select target** to choose the SD card to use, as shown in *Figure 1.24*. Then click **Continue**:

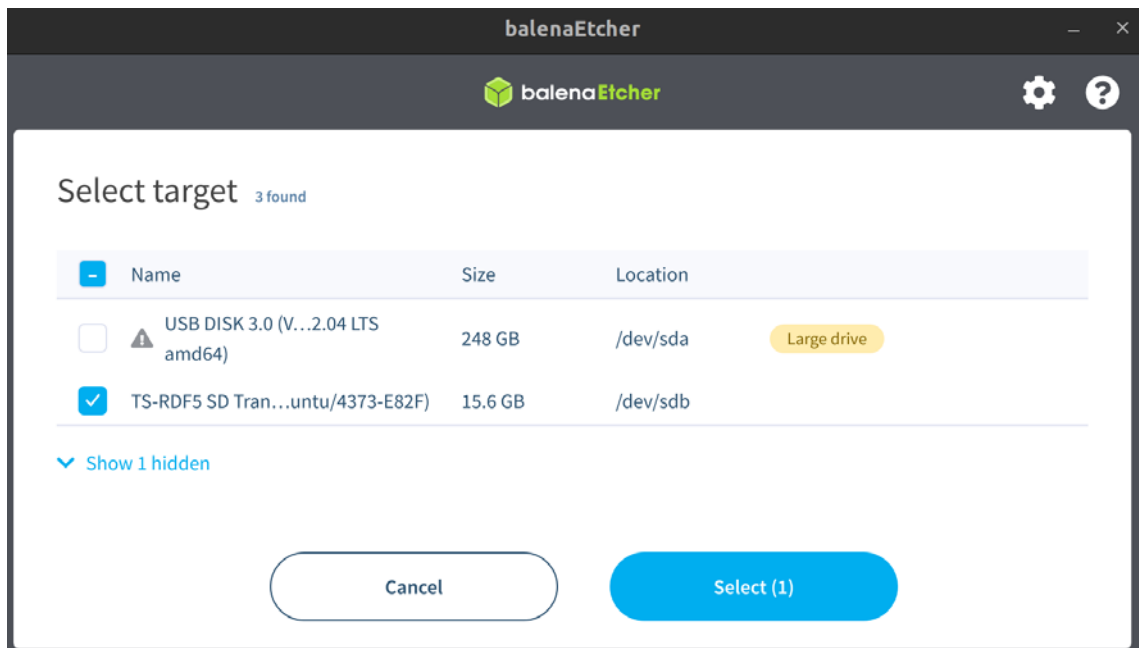


Figure 1.24: Choosing an SD card to use with Etcher for the Raspberry Pi

Once you've selected both the image file to use and the target SD card, you're ready to proceed. Click **Flash!** to begin the process of writing the image:

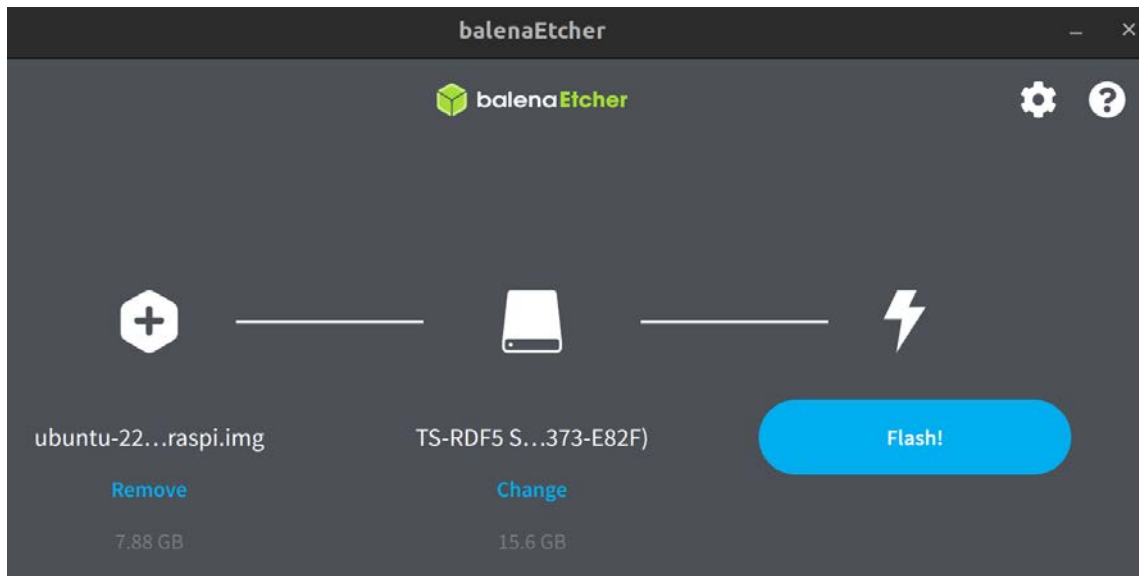


Figure 1.25: Etcher, ready to start flashing the SD card

Now, Etcher will prepare the target with the source image file. The process can take anywhere from 5 to 15 minutes depending on the speed of your hardware:

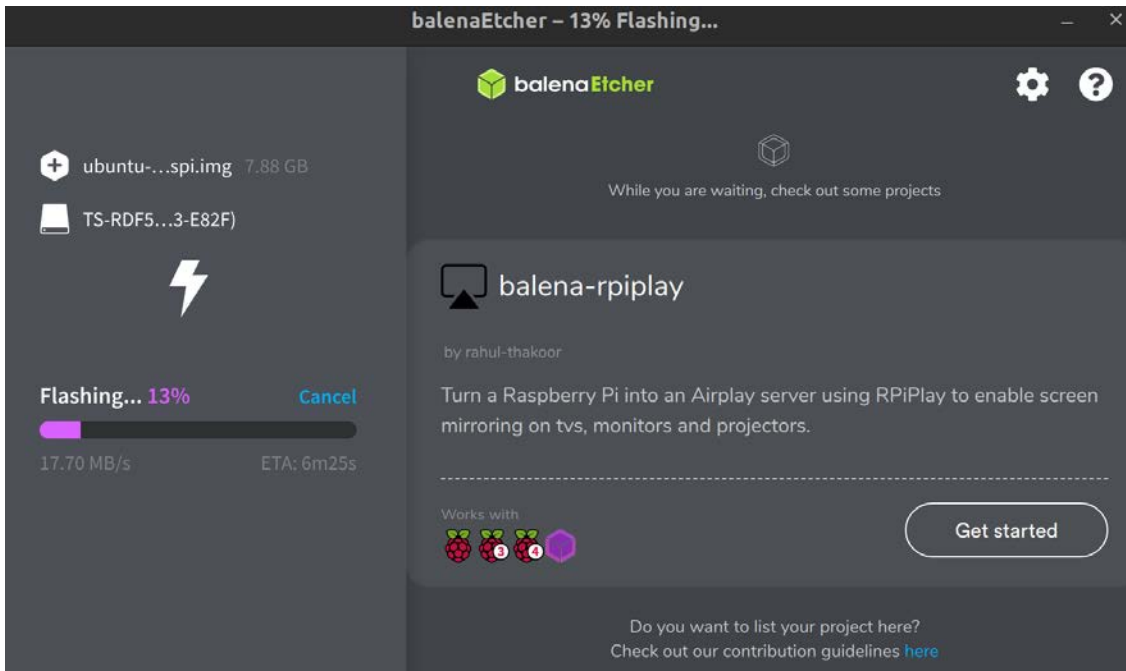


Figure 1.26: Etcher in progress

Once Etcher is finished writing to the SD card, you can eject it from your computer and insert it into the Pi, which can be accessed remotely via SSH. When it finishes booting for the first time, you can log in and begin using it as you would any other Ubuntu Server installation (the username and password both default to ubuntu). From there, you can create a new user for yourself, install applications, and experiment.

Summary

In this chapter, we covered a couple of different installation scenarios in great detail. As with most Linux distributions, Ubuntu Server is scalable and able to be installed on a variety of server types. Physical devices, VMs, and even the Raspberry Pi have versions of Ubuntu available. The process of installation was covered step by step, and you should now have an Ubuntu Server instance of your own to configure as you wish. Also in this chapter, we covered determining the role of your server, the process of creating bootable media, and a walkthrough of setting up Ubuntu Server on your server device (as well as on a Raspberry Pi). You're now on your way to Ubuntu Server mastery!

In the next chapter, we'll show you how to manage users. You'll be able to create them, delete them, change them, manage password expiration, and more. We will also cover permissions so that you can determine what your users are allowed to do on your server. See you there!

Relevant tutorials

- Install Ubuntu Server (Video Tutorial) from LearnLinuxTV:
<https://linux.video/install-ubuntu-server>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



2

Managing Users and Permissions

In the previous chapter, we set up our very own Ubuntu Server installation, and we can now learn how to maintain it, starting with a look at managing who is able to use our server.

As administrators of Ubuntu servers, users can be your greatest asset and also your biggest headache. During your career, you'll add countless new users, manage their passwords, remove their accounts when they leave the company, and grant or remove access to resources across the network. Even on servers on which you're the only user, you'll still find yourself managing user accounts, since even system processes run as users. To be successful at managing Linux servers, you'll also need to know how to manage permissions, create password policies, and limit who can execute administrative commands on the machine. In this chapter, we'll work through these concepts so that you have a clear idea of how to manage users and their resources.

In particular, we will cover:

- Understanding the purpose of users and groups
- Understanding when to use root
- Creating and removing users
- Understanding the `/etc/passwd` and `/etc/shadow` files
- Distributing default configuration files with `/etc/skel`
- Switching between users
- Managing groups
- Managing passwords and password policies
- Configuring administrator access with `sudo`
- Setting permissions on files and directories

In the first section, we will have a quick discussion about the nature of managing users.

Understanding the purpose of users and groups

When it comes to a server, users are very important—without users to serve, then there’s no real need for a server in the first place. The subject of user management itself within the world of IT is in and of itself quite vast. Entire books have been written on individual methods of authentication, and entire technologies (such as **Lightweight Directory Access Protocol**, or **LDAP**) exist around it. In this chapter, we’ll look at managing users that exist locally on our server, and the groups that help define what they are able to do.

Since Ubuntu Server is a distribution of Linux, it adopts the Unix style of managing user accounts, groups, and permissions. Although our focus is on Ubuntu, many of the same commands around user management that you’ll learn in this chapter will apply to other platforms as well. There are commands that allow you to add, remove, and change users, as well as commands that allow you to alter permissions.

Users in the context of a server refer to who (or what) is able to use the server. For example, you may have an accountant named Susan, or an IT administrator named Haneef, who both need to access the server. Perhaps Susan only needs access to a file share directory for accounting-related files, and Haneef might have more access to the server as a system administrator. The user accounts we create on our server will represent the actual people that will use it.

Groups allow us to segregate access to specific files and directories. As we’ll learn later, files and directories have user and group assignments. When combined with permissions, we’ll be able to manage what our users are able to do with our server.

Users aren’t always people, though. We also have system users on our server that applications and running processes might use for background or automated tasks. An example of this might be a backup job, and you may have a backup user that runs a task in the background to facilitate some sort of file copy task that copies important files to another place. You don’t have to worry about system-related users for now, just know that they exist. You’ll see more examples of this as we go through the book.

More advanced organizations may have a central login server, such as **Active Directory (AD)** or standard LDAP. There are others aside from those, as well. In this book, we won’t cover those technologies, but just keep in mind that central authentication servers are a possibility for your organization, should you choose to explore them.

The most powerful user of all, though, is **root**. This special user gives us the most control, but as you’ll see in the next section, that comes with risks.

Understanding when to use root

In the last chapter, we set up our very own Ubuntu Server installation. During the installation process, we were instructed to create a user account to act as a system administrator. So, at this point, we should have at least two users on our server. We have the aforementioned administrative user, as well as **root**. We can certainly create additional user accounts with varying levels of access (and we will do so in this chapter), but before we get to that, some discussion is in order regarding the administrator account you created, as well as the **root** user that was created for you.

The root user account exists on all Linux distributions and is the most powerful user account on the planet. The root user account can be used to do anything within your server, and I do mean *anything*. Want to create files and directories virtually anywhere on the filesystem? Want to install software? These processes are easily performed with root. The root account can even be used to destroy your entire installation with one typo or ill-conceived command: if you instruct root to delete all the files on your entire hard disk, it won't hesitate to do so. It's always assumed on a Linux system that if you are using root, you are doing so because you know what you are doing. So, there's often not so much as a confirmation prompt while executing any command as root. It will simply do as instructed, for better or worse.

It's for this reason that every Linux distribution I've ever used has stated, or at least highly recommended, that you create a standard user during the installation process. It's generally recommended in the Linux community for an administrator to have their own account and then switch to root whenever a task comes up that requires root privileges to complete. This approach is less likely to destroy your server with an accidental typo or bad command. Some administrators will strictly use root at all times without any issue, but again, it's recommended to use root only when you have to.

Most distributions ask you to create a root password during installation in order to protect that account. Even Debian (on which Ubuntu is based) has you set a root password during installation. Ubuntu just decides to do things a little bit differently. The reason for this is that, unlike many other distributions, Ubuntu defaults to locking out the root account altogether. There's nothing stopping you from enabling root, or switching to the root user after you log in. Being disabled by default just means the root account isn't as easily accessible as it normally would be. I'll cover how to enable this account later in this chapter, should you feel the need to do so.

An exception to this rule is that some VPS providers, such as Linode, will enable the root account even on their Ubuntu servers. Sometimes, the root password will be randomly generated and emailed to you. However, you should still create a user for yourself with administrative access regardless.

Instead of using root outright, Ubuntu (as well as its server version) recommends the use of `sudo`. Specifically, `sudo` enables you to run individual commands with elevated privileges instead of being logged in as root all the time.

Using `sudo` to run privileged commands

I'll go over how to manage `sudo` later on in this chapter, but for now, just keep in mind that the purpose of `sudo` is to enable you to use your user account to do things that normally only root would be able to do. For example, as a normal user, you cannot issue a command such as the following to install a software package (don't worry about the `apt` command for now, as we'll cover that in *Chapter 3, Managing Software Packages*):

```
apt install tmux
```

Instead, you'll receive an error:

```
E: Could not open lock file /var/lib/dpkg/lock-frontent - open (13: Permission denied)
```

```
E: Unable to acquire the dpkg frontend lock (/var/lib/dpkg/lock-frontent), are you root?
```

But if you prefix the command with `sudo` (assuming your user account has access to it), the command will work just fine:

```
sudo apt install tmux
```

When you use `sudo`, you'll be asked for your user's password for confirmation, and then the command will execute. Subsequent commands prefixed with `sudo` may not prompt for your password, as it will cache your password for a short period of time until it times out or the terminal is closed. Understanding this should clarify the usefulness of the user account you created during installation. I referred to this user as an administrative account earlier, but it's really just a user account that is able to utilize `sudo`. Ubuntu Server automatically gives the first user account you create during installation access to `sudo`.

The intent is that you'll use that account to administer the system, rather than `root`. When you create additional user accounts, they will not have access to `sudo` by default, unless you explicitly grant it to them.

Creating and removing users

Creating users in Ubuntu can be done with one of two commands: `adduser` and `useradd`. This can be a little confusing at first, because both of these commands do the same thing (in different ways) and are named very similarly. I'll go over the `useradd` command first and then I'll explain how `adduser` differs. You may even prefer the latter, but we'll get to that in a moment.

Using `useradd`

First, here's an example of the `useradd` command in action:

```
sudo useradd -d /home/jdoe -m jdoe
```

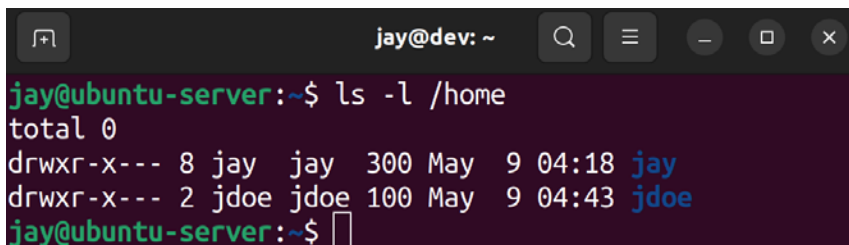
With this command, I created a user named `jdoe`. With the `-d` option, I'm clarifying that I would like a home directory created for this user, and following that, I called out `/home/jdoe` as the user's home directory. The `-m` flag tells the system that I would like the home directory to be created during the process; otherwise, I would've had to create the directory myself. Finally, I called out the username for my new user (in this case, `jdoe`).

As we go along in this book, there will be commands that require root privileges in order to execute. The preceding command was an example of this. For commands that require such permissions, I'll prefix the commands with `sudo`. When you see these, it just means that root privileges are required to run the command. For these, you can also log in as root (if root is enabled) or switch to root to execute these commands as well. However, as I mentioned before, using `sudo` instead of using the root account is strongly encouraged.

Now, list the storage of /home using the following command:

```
ls -l /home
```

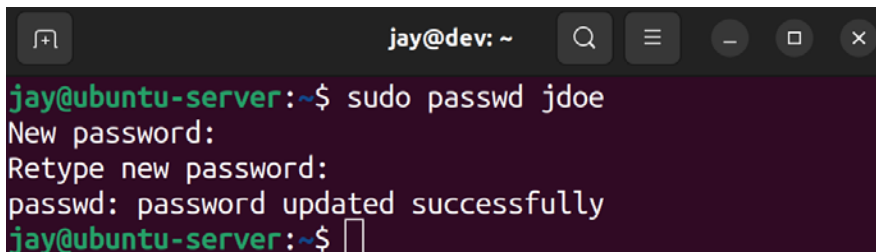
You should see a folder listed there for our new user:



```
jay@dev: ~  
jay@ubuntu-server:~$ ls -l /home  
total 0  
drwxr-x--- 8 jay jay 300 May  9 04:18 jay  
drwxr-x--- 2 jdoe jdoe 100 May  9 04:43 jdoe  
jay@ubuntu-server:~$
```

Figure 2.1: Listing the contents of /home after our first user was created

What about creating our user's password? We may have been asked for our current user's password due to using `sudo`, but we weren't asked for a password for the new user. To create a password for the user, we can use the `passwd` command. The `passwd` command defaults to allowing you to change the password for the user you're currently logged in as, but it also allows you to set a password for any other user if you run it as `root` or with `sudo`. If you enter `passwd` by itself, the command will first ask you for your current password, then your new password, and then it will ask you to confirm your new password again. If you prefix the command with `sudo` and then specify a different user account, you can set the password for any user you wish. An example of the output of this process is as follows:



```
jay@dev: ~  
jay@ubuntu-server:~$ sudo passwd jdoe  
New password:  
Retype new password:  
passwd: password updated successfully  
jay@ubuntu-server:~$
```

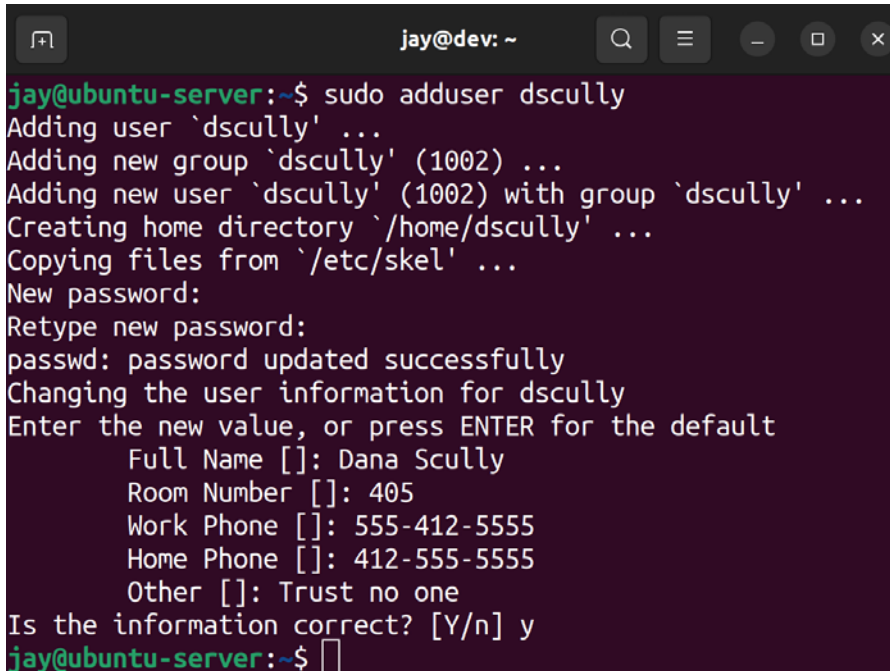
Figure 2.2: Changing the password of a user

As you can see in the previous screenshot, you won't see any asterisks or any kind of output when you type a password using the `passwd` command. This is normal. Although you won't see any visual indication of input, your input is being recognized.

Now we have a new user and we were able to set a password for that user. The `jdoe` user will now be able to access the system with the password we've chosen. This user won't have access to `sudo` by default, but we'll cover how to change this later on in the chapter.

Using adduser

Earlier, I mentioned the `adduser` command as another way of creating a user. The difference (and convenience) of this command should become apparent immediately once you've used it. Go ahead and give it a try; execute `adduser` along with a username for a user you wish to create. An example run of this process is as follows:

A terminal window titled 'jay@dev: ~' showing the execution of the 'adduser' command. The output is as follows:

```
jay@ubuntu-server:~$ sudo adduser dscully
Adding user `dscully' ...
Adding new group `dscully' (1002) ...
Adding new user `dscully' (1002) with group `dscully' ...
Creating home directory `/home/dscully' ...
Copying files from `/etc/skel' ...
New password:
Retype new password:
passwd: password updated successfully
Changing the user information for dscully
Enter the new value, or press ENTER for the default
    Full Name []: Dana Scully
    Room Number []: 405
    Work Phone []: 555-412-5555
    Home Phone []: 412-555-5555
    Other []: Trust no one
Is the information correct? [Y/n] y
jay@ubuntu-server:~$
```

Figure 2.3: Creating a user with the `adduser` command

In the preceding process, I executed `sudo adduser dscully` (commands that modify users require `sudo` or `root`) and then I was asked a series of questions regarding how I wanted the user to be created. I was asked for the password (twice), Full Name, Room Number, Work Phone, and Home Phone. In the Other field, I entered the comment `Trust no one`, which is a great mindset to adopt while managing users. The latter prompts prior to the final confirmation were all optional: I didn't have to enter Full Name, Room Number, and so on. I could've pressed *Enter* to skip those prompts if I wanted to. The only things that are really required are the username and the password.

From the output, we can see that the `adduser` command performed quite a bit of work for us. The command defaulted to using `/home/dscully` as the home directory for the user, the account was given the next available **User ID (UID)** and **Group ID (GID)** of `1002`, and it also copied files from `/etc/skel` into our new user's home directory. In fact, both the `adduser` and `useradd` commands copy files from `/etc/skel`, but `adduser` is more verbose regarding the actions it performs.

Don't worry if you don't understand what UID, GID, and `/etc/skel` are yet. We'll work through those concepts soon.

In a nutshell, the `adduser` command is much more convenient in the sense that it prompts you for various options while it creates the user without requiring that you memorize command-line options. It also gives you detailed information about what it has done. At this point, you may be wondering why someone would want to use `useradd` at all, considering how much more convenient `adduser` seems to be. Unfortunately, `adduser` is not available on all distributions of Linux. It's best to familiarize yourself with `useradd` in case you find yourself on a Linux system that's not Ubuntu.

It may be interesting for you to see what exactly the `adduser` command is. It's not even a binary program—it's a **shell script**. A shell script is simply a text file that can be executed as a program. You don't have to worry too much about scripting now, as we will cover it in *Chapter 6, Boosting Your Command-line Efficiency*. In the case of `adduser`, it's a script written in **Perl**, which is a programming language that is sometimes used for administrative tasks. Since it's not binary, you can even open it in a text editor in order to view all the code that it executes behind the scenes. However, make sure you don't open the file in a text editor with root privileges, to ensure that you don't accidentally save changes to the file and break the script. The following command will open `adduser` in a text editor on an Ubuntu Server system:

```
nano /usr/sbin/adduser
```

Use your up/down arrows as well as the *Page Up* and *Page Down* keys to scroll through the file. When you're finished, press `Ctrl+x` on your keyboard to exit the text editor. If the editor prompts you to save changes, don't do so. Anyway, those of you with keen eyes will likely notice that the `adduser` script is calling `useradd` to perform its actual work. So either way, you're either directly or indirectly using `useradd`.

Now that we know how to create users, it will be useful to understand how to remove them as well.

Removing users

Removing or disabling an account is very important when a user no longer needs to access a system, as unmanaged accounts often become a security risk. To remove a user account, we'll use the `userdel` command.

Before removing an account, though, there is one very important question you should ask yourself. Will you (or another person) need access to the user's files? Most companies have retention policies in place that detail what should happen to a user's data when they leave the organization. Sometimes, these files are copied into an archive for long-term storage. Often, a manager, coworker, or new hire will need access to the former user's files, perhaps to continue working on a project where they left off. It's important to understand this policy ahead of managing users. If you don't have a policy in place that outlines retention requirements for files when users resign, you should probably work with your management team and create one.

By default, the `userdel` command does not remove the contents of the user's home directory. Here, we use the following command to remove `dscully` from the system:

```
sudo userdel dscully
```


We can see that the files for the `dscully` user still exist by entering the following command:

```
ls -l /home
```

The preceding commands will result in the following outputs:

```

jay@dev: ~
jay@ubuntu-server:~$ sudo userdel dscully
jay@ubuntu-server:~$ ls -l /home
total 0
drwxr-x--- 2 1002 1002 100 May  9 04:53 dscully
drwxr-x--- 8 jay  jay  300 May  9 04:18 jay
drwxr-x--- 2 jdoe jdoe 100 May  9 04:43 jdoe
jay@ubuntu-server:~$

```

Figure 2.4: The home directory for the user `dscully` still exists, even though we removed the user

With the `/home` directory for `dscully` still existing, we're able to move the contents of this directory anywhere we would like to. If we had a directory called `/store/file_archive`, for example, we could easily move the files there:

```
sudo mv /home/dscully /store/file_archive
```

Of course, it's up to you to create the directory where your long-term storage will ultimately be, but you get the idea.

If you weren't already aware, you can create a new directory with the `mkdir` command. You can create a directory within any other directory that your logged-in user has access to. The following command will create the `file_archive` directory I mentioned in the preceding example:

```
sudo mkdir -p /store/file_archive
```

The `-p` flag simply creates the parent directory if it didn't already exist.

If you do actually want to remove a user's home directory at the same time that you remove an account, just add the `-r` option. This will eliminate the user and their data in one shot:

```
sudo userdel -r dscully
```

To remove the `/home` directory for the user after the account was already removed (if you didn't use the `-r` parameter the first time), use the `rm -r` command to get rid of it, as you would any other directory:

```
sudo rm -r /home/dscully
```

It probably goes without saying, but the `rm` command can be extremely dangerous. If you're logged in as root or using `sudo` while using `rm`, you can easily destroy your entire installed system if you're not careful. *DO NOT run this command*, but as a hypothetical example, the following command (while seemingly innocent at first glance) will likely completely destroy your entire filesystem:

```
sudo rm -r / home/dscully
```

Notice the typo: I accidentally typed a space after the first forward slash. I literally accidentally told my system to remove the contents of the entire filesystem. If that command were executed, the server probably wouldn't even boot the next time we attempted to start it. All user and program data would be wiped out. If there was ever any single reason for us to be protective over the root account, the `rm` command is most certainly it!

At this point, we understand how to add and remove users. In the next section, we'll look deeper into passwords.

Understanding the `/etc/passwd` and `/etc/shadow` files

Now that we know how to create (and delete) user accounts on our server, we are well on our way to being able to manage our users. But where exactly is this information stored? We know that users store their personal files in `/home`, but is there some kind of database somewhere that keeps track of which user accounts are on our system? Actually, user account information is stored in two special text files:

- `/etc/passwd`
- `/etc/shadow`

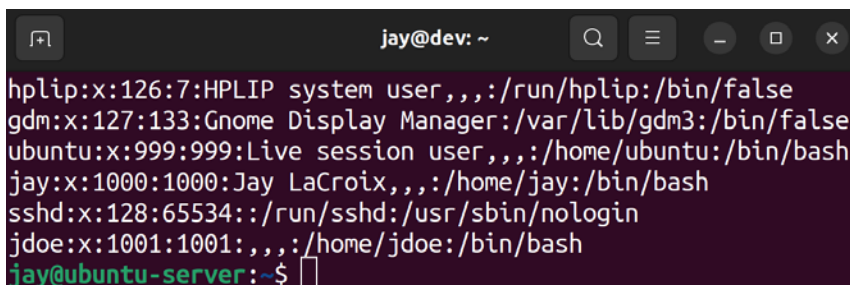
You can display the contents of each of those two files with the following commands. Note that any user can look at the contents of `/etc/passwd`, while only root has access to `/etc/shadow`:

```
cat /etc/passwd
sudo cat /etc/shadow
```

Go ahead and take a look at these two files (just don't make any changes), and I will help you understand them.

Understanding the `/etc/passwd` file

First, let's go over the `/etc/passwd` file. What follows is some example output from this file on my test server. For brevity, I have limited the output to the last eight lines:



```
hplip:x:126:7:HPLIP system user ,,,:/run/hplip:/bin/false
gdm:x:127:133:Gnome Display Manager:/var/lib/gdm3:/bin/false
ubuntu:x:999:999:Live session user ,,,:/home/ubuntu:/bin/bash
jay:x:1000:1000:Jay LaCroix ,,,:/home/jay:/bin/bash
sshd:x:128:65534::/run/sshd:/usr/sbin/nologin
jdoe:x:1001:1001:,,,:/home/jdoe:/bin/bash
jay@ubuntu-server:~$
```

Figure 2.5: Example `/etc/passwd` file

Each line within this file corresponds to a user account on the system. Entries are split into columns, separated by a colon (:). The username is in the first column, so you can see that I've created users `jay` and `jdoe`. The next column on each is simply an `x`. I'll go over what that means a bit later. For now, let's skip to the third and fourth columns, which reference the UID and GID respectively.

On a Linux system, user accounts and groups are actually referenced by their IDs. While it's easier for you and I to manage users by their names, usernames and group names are nothing more than a label placed on the UID and GID in order to help us identify them more easily.

For example, it may be frustrating to try to remember that `jdoe` is UID `1001` on our server each time we want to manage this account. Managing it by referring to the account as `jdoe` is easier for humans, since we don't remember numbers as well as we do names. But to Linux, each time we reference user `jdoe`, we're actually just referencing UID `1001`. When a user is created, the system (by default) automatically assigns the next available UID to the account. If you manage multiple Ubuntu servers, note that the UIDs will not match from one system to another, so keep in mind that UIDs don't synchronize between installations.

In my case (as shown in *Figure 2.5*), the UID of each user is the same as their GID. This is just a coincidence on my system and it isn't always that way in practice. While I'll discuss creating groups later in this chapter, understand that creating groups works in a similar way to creating users, in the sense that the group is assigned the next available GID in much the same way as new user accounts are assigned the next available UID. When you create a user, the user's primary group is the same as their username (unless you request otherwise). For example, when I created `jdoe`, the system also automatically created a `jdoe` group as well.

This is what you're actually seeing here—the UID for the user, as well as the GID for the user's primary group. Again, we'll get to groups in more detail later.

You probably also noticed that the `/etc/passwd` file on your system contains entries for many more users than the ones we've created ourselves. This is perfectly normal, as Linux uses user accounts for various processes and services that run in the background. You'll likely never interact with the default accounts at all, though you may someday create your own system user for a process to run as. For example, perhaps you'll create a data processor account for an automated data-processing script to run under.

Anyway, back to our `/etc/passwd` file. The fifth column is designated for user info, most commonly the user's first and last names. In my example, the fifth field is blank for `jdoe`, as I created `jdoe` with the `useradd` command, which didn't prompt me for the first and last names. This field is also nicknamed the GECOS field, and you may see it referred to as such when you read the documentation.

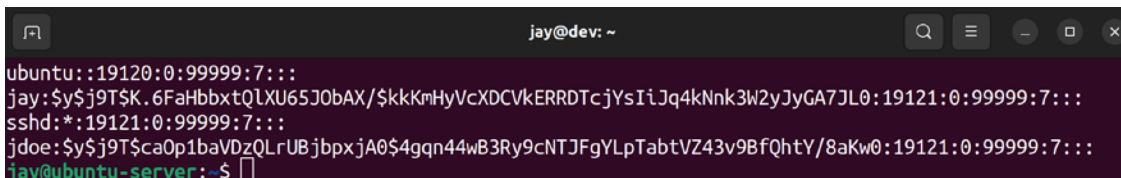
In the sixth column, the home directory for each user is shown. In the case of `jdoe`, it's set as `/home/jdoe`. Finally, we designate the user's shell as `/bin/bash`. This field refers to the default shell the user will use, which defaults to `/bin/bash` when an account is created with the `adduser` command, and `/bin/sh` when created with the `useradd` command. (If you have no preference, `/bin/bash` is the best choice for most.) If we want the user to use a different shell, we can clarify that here (though shells other than `/bin/bash` aren't covered in this book). If we wanted, we could change the user's shell to something invalid to prevent them from logging in at all. This is useful for when a security issue requires us to disable an account quickly.

Understanding the `/etc/shadow` file

With that out of the way, let's take a look at the `/etc/shadow` file. We can use `cat` to display the contents like any other text file, but unlike `/etc/passwd`, we need root privileges in order to view it. So, go ahead and display the contents of this file, and I'll walk you through it:

```
sudo cat /etc/shadow
```

This will display the following output:

A terminal window titled 'jay@dev: ~' showing the output of the command 'sudo cat /etc/shadow'. The output lists four users: 'ubuntu', 'jay', 'sshd', and 'jdoe'. Each line contains a username, a field separator, a password hash, another field separator, and a UID. The password hashes are long, alphanumeric strings. The terminal prompt is '\$' and the cursor is at the end of the last line.

```
ubuntu::19120:0:99999:7:::  
jay:$y$j9T$K.6FaHbbxtQLXU65JObAX/$kkKmHyVcXDCVkERRDtcjYsIiJq4kNnk3W2yJyGA7JL0:19121:0:99999:7:::  
sshd:*:19121:0:99999:7:::  
jdoe:$y$j9T$ca0p1baVDzQLrUBjbpXjA0$4gqn44wB3Ry9cNTJFgYLPtabtVZ43v9BfQhtY/8aKw0:19121:0:99999:7:::  
jay@ubuntu-server:~$
```

Figure 2.6: Example `/etc/shadow` file

The preceding screenshot, *Figure 2.6*, shows the last four lines of this file on my server. First, we have the username in the first column—no surprises there. Note that the output is not showing the UID for each user in this file. The system knows which username matches which UID based on the `/etc/passwd` file, so there's no need to repeat that here. In the second column, we have what appears to be gobbledygook. Actually, that's the most important part of this entire file. That's the actual hash for the user's password.

A password hash is a conversion of the actual password to a different string that represents the original password. This is a one-way conversion, so you cannot find the actual password by reverse-engineering the hash. In the `/etc/passwd` file, the hash of the password is stored rather than the actual password, for security purposes.

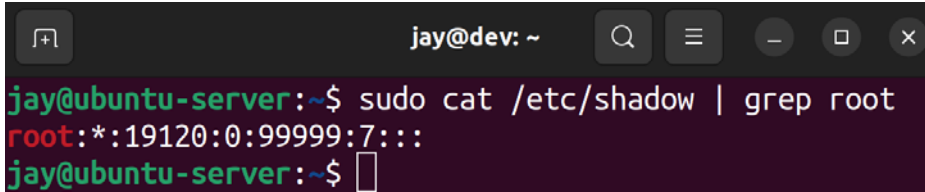
If you recall, in the `/etc/passwd` file, each user listing had an `x` for the second column, and I mentioned I would explain that later. What the `x` refers to is the fact that the user's password is encrypted and simply not stored in `/etc/passwd` and is instead stored in `/etc/shadow`. After all, the `/etc/passwd` file is viewable by everyone, so it would compromise security quite a bit if anyone could just open up the file and see what everyone's passwords were.

In the days of old, you could actually store a user's password in `/etc/passwd`, but it's never done that way anymore. Whenever you create a user account on a modern Linux system, the user's password is encrypted (an `x` is placed in the second column of `/etc/passwd` for the user), and the actual password hash is stored in the second column of `/etc/shadow` to keep it away from prying eyes. Hopefully, now the relationship between these two files has become apparent.

Remember earlier I mentioned that the root user account is locked out by default? Well, let's actually see that in action. Execute the following command to see the root user account entry in `/etc/shadow`:

```
sudo cat /etc/shadow | grep root
```

On my system, I get the following output:



```

jay@dev: ~
jay@ubuntu-server:~$ sudo cat /etc/shadow | grep root
root:*:19120:0:99999:7:::
jay@ubuntu-server:~$

```

Figure 2.7: Example `/etc/shadow` file

You should notice right away that the root user account doesn't have a password hash at all. Instead, there's an asterisk where the password hash would've been. In practice, placing an asterisk or exclamation point here is one way to lock an account. Even easier, you can use the `passwd -l` command against an account to lock it without having to edit a file. But either way, we can still switch to the root account anytime (which I'll show you how to do later on in this chapter). Entering an asterisk or exclamation mark in the second field creates the restriction that we can't directly log in as that user from the shell or over the network. We have to log in to the system as a normal user account first, and then we can still switch to that user if we want to.

With the discussion of password hashes out of the way, there are a few more fields within `/etc/shadow` entries that we should probably understand. Here's a contrived example line:

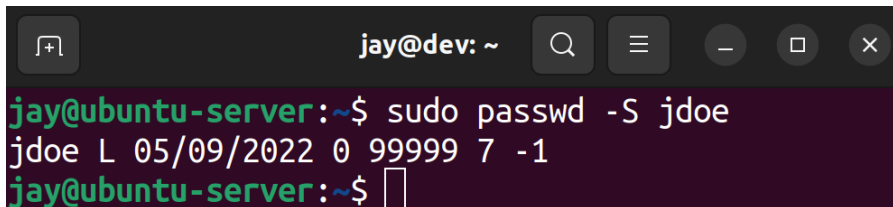
```
mulder:$6$TPxx8Z.:16809:0:99999:7:::
```

Continuing on with the third column, we can see the number of days since the **Unix epoch** that the password was last changed. For those that don't know, the Unix epoch is January 1, 1970. Therefore, we can read that column as the password having last been changed 16,809 days after the Unix epoch.

Personally, I like to use the following command to show more easily when the password was last changed:

```
sudo passwd -S <username>
```

This will result in an output that looks something like the following:



```

jay@dev: ~
jay@ubuntu-server:~$ sudo passwd -S jdoe
jdoe L 05/09/2022 0 99999 7 -1
jay@ubuntu-server:~$

```

Figure 2.8: Checking the date of the last password change for a user

By executing this command, you can view information about any account on your system. The first column is obviously the username. The second has to do with the status of the password, which in this case is L, which refers to the fact that the user has a password that is locked. It would show P if the password was set and usable, or NP if the user didn't have a password at all.

The third column of this command's output gives you the actual date of the last password change for the user. The fourth column tells us how many days are required to pass before the user will be able to change their password again. In this example, `jd` can change the password any time because the minimum number of days is set to `0`. We'll talk about how to set the minimum number of days later on in this chapter, but I'll give you a brief explanation of what this refers to. At first, it may seem silly to require a user to wait a certain number of days to be able to change their password. However, never underestimate the ability of your users to be oppositional. It's quite common for a user, when required to change their password, to change their password to satisfy history requirements, only to then just change it back to what it was originally. By setting a minimum number of days, you're forcing a waiting period in between password changes, making it less convenient for your users to cycle back through to their original password.

The fifth column, as you can probably guess, is the maximum number of days that can pass between password changes. If you require your users to change their passwords every certain number of days, you'll see that in this column. By default, this is set to `99999` days. That number of days is way beyond the human lifespan, so it may as well be infinite.

Continuing with the sixth column, we have the number of days that will elapse before the expiration date on which the user is warned that they will soon be required to change their password. In the seventh column, we set how many days can pass after the password expires, in which case the account will be disabled. With our example user, this is not set. Finally, with the eighth column (which is not visible), we can see the number of days since the Unix epoch that will elapse before the account is disabled (in our case, there's nothing here, so there is no disabled day set).

We'll go over setting these fields later, but for now, hopefully you understand the contents of the `/etc/shadow` file better.

If at any time you'd like additional clarification, feel free to check out the Ubuntu man pages. A man page (short for manual page) can give you quite a bit more information about commands as well as files. For example, the following command shows you the man page for the `ls` command:

```
man ls
```

More specific to this section, you can retrieve man pages for the `/etc/shadow` files as well:

```
man passwd
man shadow
```

Press `q` on your keyboard to exit out of a man page. Feel free to check out the man pages for any command in this book to learn more as you go along.

Now that we fully understand how to manage our users, we can also look at how to provide them with default files in their home directory.

Distributing default configuration files with `/etc/skel`

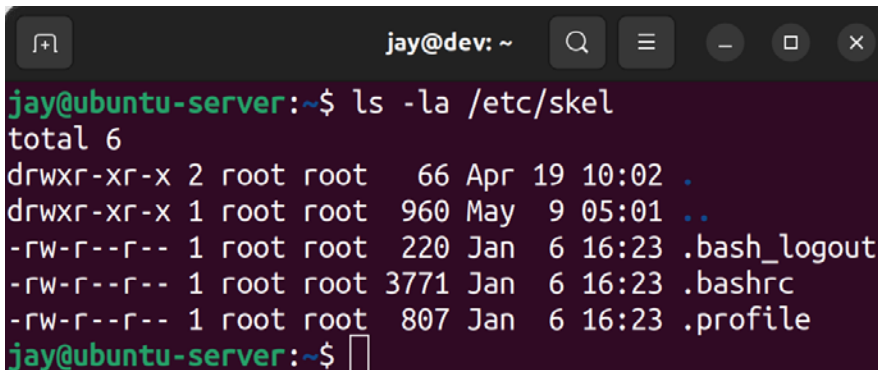
In a typical organization, there are usually some defaults that are recommended for users in terms of files and configuration.

For example, in a company that performs software development, there are likely recommended settings for text editors and version control systems. Files that are contained within `/etc/skel` are copied into the home directory for all new users when you create them (assuming you've chosen to create a home directory while setting up the user).

In fact, you can see this for yourself right now. Execute the following command:

```
ls -la /etc/skel
```

Now, you should be able to view the contents of the `/etc/skel` directory:



```
jay@dev: ~  
jay@ubuntu-server:~$ ls -la /etc/skel  
total 6  
drwxr-xr-x 2 root root 66 Apr 19 10:02 .  
drwxr-xr-x 1 root root 960 May 9 05:01 ..  
-rw-r--r-- 1 root root 220 Jan 6 16:23 .bash_logout  
-rw-r--r-- 1 root root 3771 Jan 6 16:23 .bashrc  
-rw-r--r-- 1 root root 807 Jan 6 16:23 .profile  
jay@ubuntu-server:~$
```

Figure 2.9: Default `/etc/skel` files

You probably already know how to list files within a directory, but I added the `-a` option because I wanted to view hidden files as well. The files included in `/etc/skel` by default are hidden (their filenames begin with a period). I threw in the `-l` parameter solely because it shows a long list, which I think is easier to read.

Each time you create a new user and request a home directory to be created as well, these three files, shown in *Figure 2.9*, will be copied into their home directory, along with any other files you create here. You can verify this by listing the storage of the home directories for the users you've created so far. The `.bashrc` file in one user's home directory should be the same as any other, unless they've made changes to it.

Armed with this knowledge, it should be extremely easy to create default files for new users that you create. For example, you could create a file named `welcome` with your favorite text editor and place it in `/etc/skel`. Perhaps you may create this file to contain helpful phone numbers and information for new hires in your company. The file would then be automatically copied to the new user's home directory when you create the account. The user, after logging in, would see this file in their home directory and see the information. More practically, if your company has specific editor settings that are favored for writing code, you can include those files in `/etc/skel` as well to help ensure your users are compliant. In fact, you can include default configuration files for any application your company uses.

Go ahead and give it a try. Feel free to create some random text files and then create a new user afterward, and you'll see that these files will propagate into the home directories of the new user accounts that you add to your system.

Now that we have multiple users and have also seen how to manage their default files, we can take a look at how to switch from one user to another.

Switching users

Now that we have several users on our system, we need to know how to switch between them. Of course, you can always just log in to the server as one of the users, but you can actually switch to any user account at any time, provided you either know that user's password or have sudo access.

The command you will use to switch from one user to another is the `su` command. If you enter `su` with no options, it will assume that you want to switch to root and will ask you for the root password. As I mentioned earlier, Ubuntu locks the root account by default, so at this point you may not have a root password.

Even though Ubuntu doesn't create a password for root by default, some **Virtual Private Server (VPS)** providers unlock the root password and actually have you log in as the root user. Having an unlocked root account is not a standard Ubuntu practice, and is a customization specific to some cloud providers.

Unlocking the root account is actually really simple; all you have to do is create a root password. To do that, you can execute the following command as any user with sudo access:

```
sudo passwd
```

The command will ask you to create and confirm your root password. From this point on, you will be able to use the root account as any other account. You can log in as root or switch to root—it's fully available now. However, you really don't have to unlock the root account in order to use it. You certainly can, but there are ways to switch to root without unlocking it, and it's typically better to leave the root account locked unless you have a very specific reason to unlock it. The following command will allow you to switch to root from a user account that has sudo access:

```
sudo su -
```

Now you'll be logged in as root and will be able to execute any command you want with no restrictions whatsoever. To return to your previously logged-in account, simply type `exit`. You can tell which user you're logged in as by the value at the beginning of your bash prompt.

What if you want to switch to an account other than root? Of course, you can simply log out and then log in as that user. But you really don't have to do that. The following command will do the job, providing you know the password for the account:

```
su - <username>
```

The shell will ask for that user's password and then you'll be logged in as that user. Again, type `exit` when you're done using the account, which will return you to the one you were using before you switched.

That command is all well and good if you know the user's password, but you often won't. Typically, in an enterprise, you'll create an account, force the user to change their password at first login, and then you will no longer know that user's password.

Since you have root and sudo access, you could always change their password and then log in as them. But they'll know something is amiss if their password suddenly stops working—you're not eavesdropping, are you? Armed with sudo access, you can use sudo to change to any user you want to, even if you don't know their password. Just prefix our previous command with sudo and you'll only need to enter the password for your user account, instead of theirs:

```
sudo su - <username>
```

Switching to another user account is often very helpful for support (especially while troubleshooting permissions). As an example, say that a user comes to you complaining that they cannot access the contents of a specific directory, or they are unable to run a command. In that case, you can log in to the server, switch to their user account, and try to reproduce their problem. That way, you can not only see their problem yourself, but you can also test out whether or not your fix has solved their issue before you report back to them.

Now we have a full understanding of user accounts, and even how to switch between them. In the next section, we'll look into groups, which allow us to categorize our users.

Managing groups

Now that we understand how to create, manage, and switch between user accounts, we'll need to understand how to manage **groups** as well. The concept of groups in Linux is not very different from other platforms and pretty much serves the exact same purpose. With groups, you can more efficiently control a user's access to resources on your server. By assigning a group to a resource (a file, a directory, and so on), you can allow and disallow access to users by simply adding them or removing them from the group.

The way this works in Linux is that every file or directory has both a user and a group that takes ownership of it. This is contrary to platforms such as Windows, which can have multiple groups assigned to a single resource. With Linux, it's just one-to-one ownership: just one user and just one group assigned to each file or directory. If you list the contents of a directory on a Linux system, you can see this for yourself:

```
ls -l
```

The following is a sample line of output from a directory on one of my servers:

```
-rw-r--r-- 1 root bind 490 2022-04-15 22:05 named.conf
```

In this case, we can see that root owns the file and that the group bind is also assigned to it. Ignore the other fields for now; I'll explain them later when we get to the section of this chapter dedicated to permissions. For now, just keep in mind that one user and one group are assigned to each file or directory.

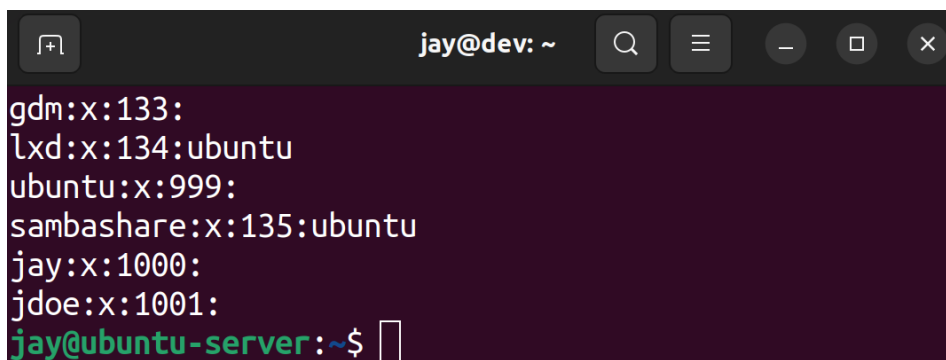
While each file or directory can only have one group assignment, any user account can be a member of any number of groups. Entering the groups command by itself with no options will tell you what groups your logged-in user is currently a member of. If you add a username to the groups command, you'll see which groups that user is a member of. Go ahead and give the groups command a try with and without providing a username to get the idea.

On the Ubuntu Server platform, you'll likely see that each of your user accounts is a member of a group that's named the same as your username. As I mentioned earlier, when you create a user account, you're also creating a group with the same name as the user. On some Linux distributions, though, a user's primary group will default to a group called `users` instead. If you were to execute the `groups` command as a user on the Ubuntu desktop platform, you would likely see additional groups. This is due to the fact that distributions of Linux that cater to being a server platform are often more stripped down and users on desktop platforms need access to more objects such as printers, audio cards, and so on. Some packages that can be installed also add additional system users to the server.

If you were curious as to which groups exist on your server, all you would need to do is `cat` the contents of the `/etc/group` file. Similar to the `/etc/passwd` file we covered earlier, the `/etc/group` file contains information regarding the groups that have been created on your system. Go ahead and take a look at this file on your system:

```
cat /etc/group
```

The following is sample output from this file on one of my servers:



```
jay@dev: ~  
gdm:x:133:  
lxd:x:134:ubuntu  
ubuntu:x:999:  
sambashare:x:135:ubuntu  
jay:x:1000:  
jdoe:x:1001:  
jay@ubuntu-server:~$
```

Figure 2.10: Sample output from the `/etc/group` file

Like before, the columns in this file are separated by colons, though each line is only four columns long. In the first column, we have the name of the group. No surprise there. In the second, we are able to store a password for the group, but this is not used often as it's actually a security risk to do so. In the third column, we have the GID, which is similar in concept to the UID from when we were discussing users. Finally, in the last column, we (would) see a comma-separated list of each user that is a member of each of the groups.

Several entries don't show any group memberships at all. Each user is indeed a member of their own group, so this is implied even though it doesn't explicitly call that out in this file. If you take a look at the `/etc/passwd` entries for your users, you will see that their primary group (shown as the third column in the form of a GID) references a group contained in the `/etc/group` file.

Creating new groups on your system is easy to do and is a great idea for categorizing your users and what they are able to do. Perhaps you create an `accounting` group for your accountants, an `admins` group for those in your IT department, and a `sales` group for your salespeople. The `groupadd` command allows you to create new groups.

If you wanted to, you could just edit the `/etc/group` file and add a new line with your group information manually, although, in my opinion, using `groupadd` saves you some work and ensures that group entries are created properly. Editing group and user files directly is typically frowned upon (and a typo can cause serious problems). Anyway, what follows is an example of creating a new group with the `groupadd` command:

```
sudo groupadd admins
```

If you take a look at the `/etc/group` file again after adding a new group, you'll see that a new line was created in the file and a GID was chosen for you (the first one that hadn't been used yet). Removing a group is just as easy. Just issue the `groupdel` command followed by the name of the group you wish to remove:

```
sudo groupdel admins
```

Next, we'll take a look at the `usermod` command, which will allow you to actually associate users with groups. The `usermod` command is more or less a Swiss Army knife; there are several things you can do with that command (adding a user to a group is just one of its abilities). If we wanted to add a user to our `admins` group, we would issue the following command:

```
sudo usermod -aG admins myuser
```

In that example, we're supplying the `-a` option, which means append, and immediately following that, we're using `-G`, which means we would like to modify secondary group membership. I put the two options together with a single dash (`-aG`), but you could also issue them separately (`-a -G`). The example I gave only adds the user to additional groups; it doesn't replace their primary group.

Be careful not to miss the `-a` option here, as by doing so, you will instead replace all current group memberships with the new one, which is usually not what you want. The `-a` option means append, or to add the existing list of group memberships for that user.

If you wanted to change a user's primary group, you would use the `-g` option instead (lowercase `g` instead of an uppercase `G` as we used earlier):

```
sudo usermod -g <group-name> <username>
```

Feel free to check out the man pages for the `usermod` command, to see all the nifty things it allows you to manage relating to your users. You can peruse the man page for the `usermod` command with the following command:

```
man usermod
```

One additional example is changing a user's `/home` directory. Suppose that one of your users has undergone a name change, so you'd like to change their username, as well as moving their previous home directory (and their files) to a new one. The following commands will take care of that:

```
sudo usermod -d /home/jsmith jdoe -m  
sudo usermod -l jsmith jdoe
```

In that example, we're moving the home directory for `jdoe` to `/home/jsmith`, and then in the second example, we're changing the username from `jdoe` to `jsmith`.

If you wish to remove a user from a group, you can use the `gpasswd` command to do so. `gpasswd -d` will do the trick:

```
sudo gpasswd -d <username> <grouptoremove>
```

In fact, `gpasswd` can also be used in place of `usermod` to add a user to a group:

```
sudo gpasswd -a <username> <group>
```

So, now you know how to manage groups. With the efficient management of groups, you'll be able to manage the resources on your server better. Of course, groups are relatively useless without some explanation of how to manage permissions (otherwise, nothing would actually allow a member of a group access to a resource). Later on in this chapter, we'll cover permissions so that you have a complete understanding of how to manage user access.

Managing passwords and password policies

In this chapter, we've already covered a bit of password management, since I've given you a few examples of the `passwd` command. If you recall, the `passwd` command allows us to change the password of the currently logged-in user. In addition, using `passwd` as root (and supplying a username) allows us to change the password for any user account on our system. But that's not all this command can do.

Locking and unlocking user accounts

One thing I've neglected to mention regarding the `passwd` command is the fact that you can use it to lock and unlock a user's account. There are many reasons why you may want to do this. For instance, if a user is going on vacation or extended leave, perhaps you'd want to lock their account so that it cannot be used while they are away. After all, the fewer active accounts, the smaller your attack surface. To lock an account, use the `-l` option:

```
sudo passwd -l <username>
```

And to unlock it, use the `-u` option:

```
sudo passwd -u <username>
```

However, locking an account will not prevent the user from logging in if they have access to the server via SSH while utilizing public key authentication. In that case, you'd want to remove their ability to use SSH as well. One common way of doing this is to limit SSH access to users who are members of a specific group. When you lock an account, simply remove them from the group. Don't worry so much about the SSH portion of this discussion if this is new to you. We will discuss securing your SSH server in *Chapter 21, Securing Your Server*. For now, just keep in mind that you can use `passwd` to lock or unlock accounts, and if you utilize SSH, you'll want to lock your users out of that to prevent them from logging in.

However, there's more to password management than the `passwd` command, as we can also implement our own policies, such as viewing or adjusting password expiration details.

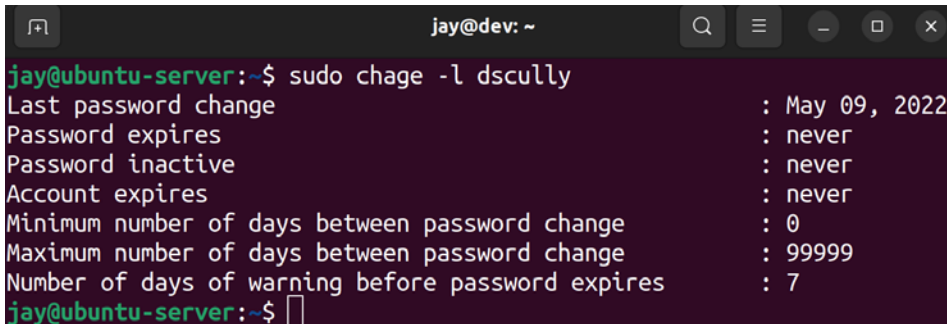
Setting password expiration information

Earlier, I mentioned that you can set an expiration date on a user's password (during our discussion on the `/etc/shadow` file). In this section, we'll go through how to actually do that. Specifically, the `chage` command gives us this ability. We can use `chage` to alter the expiration period of a user's password, but it's also a more convenient way of viewing current expiration information than viewing the `/etc/shadow` file. With the `-l` option of `chage`, along with providing a username, we can see the relevant info:

```
sudo chage -l <username>
```

Using `sudo` or `root` is not required to run `chage`. You're able to view expiration information for your own username without needing to escalate permissions. However, if you want to view information via `chage` for any user account other than your own, you will need to use `sudo`.

In the example that follows, we can see the output of this command from a sample user account:

A terminal window with a dark purple background. The prompt is 'jay@ubuntu-server:~\$'. The command 'sudo chage -l dscully' has been entered. The output is as follows:

```
jay@ubuntu-server:~$ sudo chage -l dscully
Last password change                : May 09, 2022
Password expires                    : never
Password inactive                   : never
Account expires                     : never
Minimum number of days between password change : 0
Maximum number of days between password change : 99999
Number of days of warning before password expires : 7
jay@ubuntu-server:~$
```

Figure 2.11: Output from the `chage` command

In the output, we can see values for the date of expiration, the maximum number of days between password changes, and so on. Basically, it's the same information stored in `/etc/shadow` but it's much easier to read.

If you would like to change this information, `chage` will again be the tool of choice. The first example I'll provide is a very common one. When creating user accounts, you'll certainly want them to change their password when they first log in.

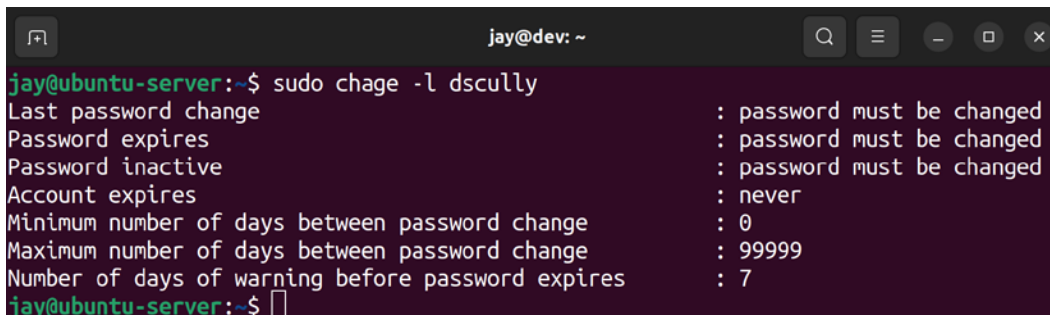
Unfortunately, not everyone will be keen on doing so. The `chage` command allows you to force a password change for a user when they first log in. Basically, you can set their number of days to expiry to `0` as follows:

```
sudo chage -d 0 <username>
```

You can see the results of this command immediately if you run `chage -l` again against the user account you just modified:

```
sudo chage -l <username>
```

The output will display information regarding the password change:



```

jay@dev: ~
jay@ubuntu-server:~$ sudo chage -l dscully
Last password change                : password must be changed
Password expires                     : password must be changed
Password inactive                   : password must be changed
Account expires                     : never
Minimum number of days between password change : 0
Maximum number of days between password change : 99999
Number of days of warning before password expires : 7
jay@ubuntu-server:~$

```

Figure 2.12: The `chage` command listing a user that has a required password change period set

To set a user account to require a password change after a certain period of days, the following will do the trick:

```
sudo chage -M 90 <username>
```

In that example, I'm setting the user account to expire and require a password change in 90 days. When the impending date reaches 7 days before the password is to be changed, the user will see a warning message when they log in.

As I mentioned earlier, users will often do whatever they can to cheat password requirements and may try to change their password back to what it was originally after satisfying the initially required password change. You can set the minimum number of days with the `-m` flag, as you can see in the next example where we set it to 5 days:

```
sudo chage -m 5 dscully
```

The trick with setting a minimum password age is to set it so that it will be inconvenient for the user to change their password to the original one, but you still want a user to be able to change their password when they feel the need to (so don't set it too long, either). If a user wants to change their password before the minimum number of days elapses (for example, if your user feels that their account may have been compromised), they can always have you change it for them. However, if you make your password requirements too much of an inconvenience for your users, it can also work against you.

Setting a password policy

Next, we should discuss setting a password policy. After all, forcing your users to change their passwords does little good if they change it to something simple, such as `abc123`. A password policy allows you to force requirements on your users for things such as length, complexity, and so on.

To facilitate this, we have **Pluggable Authentication Module (PAM)** at our disposal. PAM gives us additional functionality and control over the process of authentication, and also provides additional plugins we can use to extend authentication and add additional features. Although a full walkthrough of PAM isn't covered in this book, I recommend keeping it fresh in your mind in case you want to add additional features later.

Specific to the subject of setting up a password policy, we can install a PAM module to enable this, which involves installing a new package:

```
sudo apt install libpam-cracklib
```

Next, let's take a look at the following file, which is provided with Ubuntu. Feel free to open it with a text editor, such as nano, as we'll need to edit it:

```
sudo nano /etc/pam.d/common-password
```

An extremely important tip while modifying configuration files related to authentication (such as password requirements, sudo access, SSH, and so on) is to always keep a root shell open at all times while you make changes, and in another shell, test those changes. Do not log out of your initial root window until you are 100% certain that your changes have been thoroughly tested. While testing a policy, make sure that not only can your users log in, but your admins too. Otherwise, you may remove your ability to log in to a server and make changes.

To enable a history requirement for your passwords (meaning the system remembers the last several passwords a user has used, preventing them from reusing them), we can add the following line to the file:

```
password required pam_pwhistory.so remember=99 use_authok
```

In the example config line, I'm using `remember=99`, which (as you can probably guess) will cause our system to remember the last 99 passwords for each user and prevent them from using those passwords again. If you've configured a minimum password age earlier, for example, 5 days, it would take the user 495 days to cycle back to their original password if you take into account that the user changes their password once every 5 days, 99 times. That pretty much makes it impossible for the user to utilize their old passwords.

Another field worth mentioning within the `/etc/pam.d/common-password` file is the section that reads `difok=3`. This configuration mandates that at least three characters have to be different before the password is considered acceptable. Otherwise, the password would be deemed too similar to the old one and refused. You can change this value to whatever you like; the default is normally 5 but Ubuntu defaults it to 3 in their implementation of this config file. In addition, you'll also see `obscure` mentioned in the file as well, which prevents simple passwords from being used (such as common dictionary words and so on).

Setting a password policy is a great practice to increase the security of your server. However, it's also important to not get carried away. In order to strike a balance between security and user frustration, the challenge is always to create enough restrictions to increase your security, while trying to minimize the frustration of your users. Of course, the mere mention of the word "password" to a typical user is enough to frustrate them, so you can't please everyone. But in terms of overall system security, I'm sure your users will appreciate the fact that they can be reasonably sure that you as an administrator have taken the necessary precautions to keep their (and your company's) data safe. When it comes down to it, use your best judgment.

Since we're on the subject of security, we should also take a look at configuring sudo itself, which we'll take care of in the next section.

Configuring administrator access with sudo

By now, we've already used sudo quite a few times in this book. At this point, you should already be aware of the fact that sudo allows you to execute commands as if you were logged in as another user, with root being the default. However, we haven't had any formal discussion about it yet, nor have we discussed how to actually modify which of your user accounts are able to utilize sudo.

On all Linux systems, you should protect your root account with a strong password and limit it to being used by as few people as possible. On Ubuntu, the root account is locked anyway, so unless you unlocked it by setting a password (or you're using a version of Ubuntu supplied by a VPS provider), it cannot be used to log in to the system. Using sudo is an alternative to logging in as root to execute commands directly, so you can give your administrators access to perform tasks that require root privileges with sudo without actually giving them your root password or unlocking the root account. In fact, sudo allows you to be a bit more granular. Using root directly is basically all or nothing—if someone knows the root password and the root account is enabled, that person is not limited and can do whatever they want. With sudo, that can also be true, but you can actually restrict some users to use only particular commands and therefore limit the scope of what they are able to do on the system. For example, you could give an admin access to install software updates but not allow them to reboot the server.

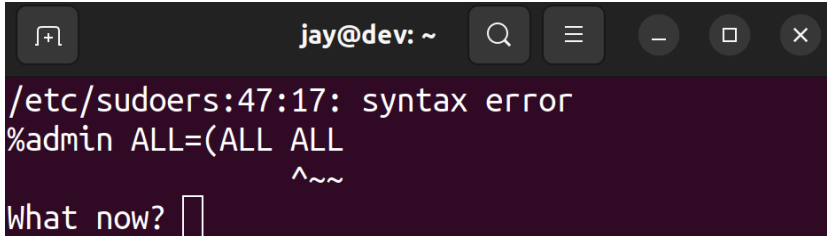
By default, members of the sudo group are able to use sudo without any restrictions. Basically, members of this group can do anything root can do (which is everything). During installation, the user account you created was made a member of sudo. To give additional users access to sudo, all you would need to do is add them to the sudo group:

```
sudo usermod -aG sudo <username>
```

Not all distributions utilize the sudo group by default, or even automatically install sudo. Other distributions require you to install sudo manually and may use another group (such as wheel) to govern access to sudo.

But again, that gives those users access to everything, and that may or may not be what you want. To actually configure sudo, we use the visudo command. This command assists you with editing /etc/sudoers, which is the configuration file that governs sudo access. Although you can edit /etc/sudoers yourself with a text editor, configuring sudo in that way is strongly discouraged. The visudo command checks to make sure your changes follow the correct syntax and helps prevent you from accidentally destroying the file. This is a very good approach, because if you did make any errors in the /etc/sudoers file, you may wind up in a situation where no one is able to gain administrative control over the server. And while there are ways to recover from such a mistake, it's certainly not a very pleasant situation to find yourself in! So, the takeaway here is never to edit the /etc/sudoers file directly; always use visudo to do so.

Here's an example of the type of warning the `visudo` command shows when you make a mistake:



```

jay@dev: ~
/etc/sudoers:47:17: syntax error
%admin ALL=(ALL ALL
                ^~~
What now? █

```

Figure 2.13: The `visudo` command showing an error

If you do see this error, press `e` to return to edit the file, and then correct the mistake.

The way this works is when you run `visudo` from your shell, you are brought into a text editor with the `/etc/sudoers` file opened up. You can then make changes to the file and save it like you would any other text file. By default, Ubuntu opens up the nano text editor when you use `visudo`. With nano, you can save changes using `Ctrl + w`, and you can exit the text editor with `Ctrl + x`.

So `visudo` allows you to make changes to who is able to access sudo. But how do you actually make these changes? Go ahead and scroll through the `/etc/sudoers` file that `visudo` opens and you should see a line similar to the following:

```
%sudo    ALL=(ALL:ALL) ALL
```

This is the line of configuration that enables sudo access to anyone who is a member of the `sudo` group. You can change the group name to any that you'd like, for example, perhaps you'd like to create a group called `admins` instead. If you do change this, make sure that you actually create that group and add yourself and your staff to be members of it before you edit the `/etc/sudoers` file or log off; it would be rather embarrassing if you found yourself locked out of administrator access to the server.

Of course, you don't have to enable access by group. You can actually call out a username instead. With the `/etc/sudoers` file, groups are preceded by `%`, while users are not. As an example of this, we also have the following line in the file:

```
root     ALL=(ALL:ALL) ALL
```

Here, we're calling out a username (in this case, `root`), but the rest of the line is the same as the one I pointed out before. While you can certainly copy this line and paste it one or more times (substituting `root` for a different username) to grant access to others, using the group approach is really the best way. It's easier to add and remove users from a group (such as the `sudo` group) than it is to use `visudo` each time.

So, at this point, you're probably wondering what the options on `/etc/sudoers` configuration lines actually mean. So far, both examples used `ALL=(ALL:ALL) ALL`. In order to fully understand sudo, understanding the other fields is extremely important, so let's go through them (using the `root` line again as an example).

The first ALL means that root is able to use sudo from any terminal. The second ALL means that root can use sudo to impersonate any other user. The third ALL means that root can impersonate any other group. Finally, the last ALL refers to what commands this user is able to use; in this case, any command they wish.

To help drive this home, I'll give some additional examples. Here's a hypothetical example:

```
charlie    ALL=(ALL:ALL) /sbin/reboot,/sbin/shutdown
```

Here, we're allowing user charlie to execute the reboot and shutdown commands. If user charlie tries to do something else (such as install a package), they will receive an error message:

```
Sorry, user charlie is not allowed to execute '/usr/bin/apt install tmux' as root on ubuntu.
```

However, if charlie wants to use the reboot or shutdown commands on the server, they will be able to do so because we explicitly called out those commands while setting up this user's sudo access. We can limit this further by changing the first ALL to a machine name, in this case, ubuntu, to reference the host name of the server I'm using for my examples. I've also changed the command that charlie is allowed to run:

```
charlie    ubuntu=(ALL:ALL) /usr/bin/apt
```

It's always a good idea to use full paths to commands when editing sudo permissions, rather than the shortened versions. For example, we used `/usr/bin/apt` here, instead of just `apt`. This is important, as the user could create a script named `apt` to do mischievous things that we normally wouldn't allow them to do. By using the full path, we're limiting the user to the binary stored at that path.

Now, charlie is only able to use `apt`. They can use `apt update`, `apt dist-upgrade`, and any other sub-command of `apt`. But if they try to reboot the server, remove protected files, add users, or anything else we haven't explicitly set, they will be prevented from doing so.

We have another problem, though. We're allowing charlie to impersonate other users. This may not be completely terrible given the context of installing packages (impersonating another user would be useless unless that user also has access to install packages), but it's bad form to allow this unless we really need to. In this case, we could just remove the `(ALL:ALL)` from the line altogether to prevent charlie from using the `-u` option of `sudo` to run commands as other users:

```
charlie    ubuntu= /usr/bin/apt
```

On the other hand, if we actually do want charlie to be able to impersonate other users (but only specific users), we can call out the username and group that charlie is allowed to act on behalf of by setting those values:

```
charlie    ubuntu=(dscully:admins) ALL
```

In that example, charlie is able to run commands on behalf of the user `dscully` and the group `admins`.

Of course, there is much more to `sudo` than what I've mentioned in this section. Entire books could be written about `sudo` (and have been), but 99% of what you will need for your daily management of this tool involves how to add access to users while being specific about what each user is able to do. As a best practice, use groups when you can (for example, you could have an `apt` group, a `reboot` group, and so on) and be as specific as you can regarding who is able to do what. This way, you're not only able to keep the root account private (or even better, disabled), but you also have more accountability on your servers.

Now that we've explored granting access to `sudo`, we will next take a look at permissions, which give us even more control over what our users are able to access.

Setting permissions on files and directories

In this section, all the user management we've done in this chapter so far all comes together. We've learned how to add accounts, manage accounts, and secure them, but we haven't actually done any work regarding managing the resources as far as who is able to access them. In this section, I'll give you a brief overview of how permissions work in Ubuntu Server and then I'll provide some examples for customizing them.

Viewing permissions

I'm sure by now that you understand how to list the contents of a directory with the `ls` command. When it comes to viewing permissions, the `-l` flag is especially handy, as the output that the long listing provides allows us to view the permissions of an object:

```
ls -l
```

The following are some example, hypothetical file listings:

```
-rw-rw-rw- 1 doctor 5 Jan 11 12:52 welcome
-rw-r--r-- 1 root root 665 Feb 19 profile
-rwxr-xr-x 1 dalek dalek 35125 Nov 7 exterminate
```

In each line, we see several fields of information. The first column is our permission string for the object (for example, `-rw-r--r--`), which we'll cover in more detail shortly. We also see the link count for the object (second column). Links are beyond the scope of this chapter but will be discussed in *Chapter 5, Managing Files and Directories*. Continuing on, the user that owns the file is displayed in the third column, the group that owns the file is in the fourth column, the size in bytes is in the fifth, the last date the file was modified is in the sixth, and finally there is the name of the file.

Keep in mind that depending on how your shell is configured, your output may look different and the fields may be in different places. For the sake of our discussion on permissions, what we're really concerned with is the permissions string, as well as the owning user and group. In this case, we can see that the first file (named `welcome`) is owned by a user named `doctor`. The second file is named `profile` and is owned by `root`. Finally, we have a file named `exterminate` owned by a user named `dalek`.

With these files, we have the permission strings of `-rw-rw-rw-`, `-rw-r--r--`, and `-rwxr-xr-x` respectively. If you haven't worked with permissions before, these may seem strange, but it's actually quite easy when you break them down. Each permission string can be broken down into four groups, as I'll show you in the following table:

Object type	User	Group	World
-	rw-	rw-	rw-
-	rw-	r--	r--
-	rwx	rwx	r-x

I've broken down each of the three example permission strings into four groups. Basically, I split them each at the first character and then again every third. The first section of a permission string is just one character. In each of these examples, it's just a single hyphen. This refers to what type the object is. Is it a directory? A file? A link? In our case, each of these permission strings is a file, because the first positions of the permission strings are all hyphens. If the object were a directory, the first character would've been a `d` instead of a `-`. If the object were a link, this field would've been `l` (lowercase *L*) instead.

In the next section, in the second column of each object, we have three characters, `rw-`, `rw-`, and `rwx` respectively. This refers to the permissions that apply to the user that owns the file. For example, here is the first permission string again:

```
-rw-rw-rw- 1 doctor doctor 5 Jan 11 12:52 welcome
```

The third section of the preceding code output shows us that `doctor` is the user that owns the file. Therefore, referring back to the table, the second column of the permission string (`rw-`) applies specifically to the user `doctor`. Moving on, the third column of permissions is also three characters; in this case, `rw-` again. This section of the permissions string refers to the group that owns the file. In this case, the group is also named `doctor`, as you can see in column four of the preceding code output. Finally, the last section of the permission string, visualized in the table (`rw-` yet again, in this case), refers to `world`, also known as *other*. This basically refers to anyone else other than the owning user and owning group. Therefore, literally everyone else gets at least `rw-` permissions on the object.

Individually, `r` stands for *read* and `w` stands for *write*. Therefore, we can read the second column (`rw-`), indicating that the user (`doctor`) has access to read and write to this file. The third column (`rw-` again) tells us the `doctor` group also has read and write access to this file. The fourth column of the permission string is the same, so anyone else would also have read and write permissions to the file as well.

The third permission string I gave as an example looks a bit different. Here it is again:

```
-rwxr-xr-x 1 dalek dalek 35125 Nov 7 exterminate
```

Here, we see the `x` attribute set. The `x` attribute refers to the ability to execute the file as a script. So, with that in mind, we know that this file is likely a script and is executable by users, groups, and others. Given the filename of `exterminate`, this is rather suspicious, and if it were a real file, we'd probably want to look into it.

If a permission is not set, it will simply be a single hyphen where there would normally be `r`, `w`, or `x`. This is the same as indicating that a permission is disabled. Here are some examples:

- `rwX`: Object has read, write, and execute permissions set for this field
- `r-x`: Object has read enabled, write disabled, and execute enabled for this field
- `r--`: Object has read enabled, write disabled, and execute disabled for this field
- `---`: Object has no permissions enabled for this field

Bringing this discussion all the way home, here are a few more permission strings:

```
-rw-r--r-- 1 sue accounting 35125 Nov 7 budget.txt
drwxr-x--- 1 bob sales 35125 Nov 7 annual_projects
```

For the first of these examples, we see that `sue` is the owning user of `budget.txt` and that this file is assigned an `accounting` group. This object is readable and writable by `sue` and readable by everyone else (group and world). This is probably bad, considering this is a budget file and is probably confidential. We'll change it later.

The `annual_projects` object is a directory, which we can tell from the `d` in the first column. This directory is owned by the `bob` user and the `sales` group. However, since this is a directory, each of the permission bits has different meanings. In the following two tables, I'll outline the meanings of these bits for files and again for directories:

- Files:

Bit	Meaning
<code>r</code>	The file can be read
<code>w</code>	The file can be written to
<code>x</code>	The file can be executed as a program

- Directories:

Bit	Meaning
<code>r</code>	The contents of the directory can be viewed
<code>w</code>	The contents of the directory can be altered
<code>x</code>	The user or group can use <code>cd</code> to go inside the directory

As you can see, permissions are read differently depending on their context: whether they apply to a file or a directory. In the example of the `annual_projects` directory, `bob` has `rwX` permissions to the directory. This means that the user `bob` can do everything (view the contents, add or remove contents, and use `cd` to move the current directory of his shell into the directory). Regarding a group, members of the `sales` group are able to view the contents of this directory and `cd` into it. However, no one in the `sales` group can add or remove items to or from the directory. On this object, *other* has no permissions set at all. This means that no one else can do anything at all with this object, not even view its contents.

Changing permissions

So, now we understand how to read permissions on files and directories. That's great, but how do we alter them? As I mentioned earlier, the `budget.txt` file is readable by everyone (*other*). This is not good because the file is confidential. To change permissions on an object, we will use the `chmod` command. This command allows us to alter the permissions of files and directories in a few different ways.

First, we can simply remove read access from the `sue` user's budget file by removing the read bit from the other field. We can do that with the following example:

```
chmod o-r budget.txt
```

If we are currently not in the directory where the file resides, we need to give a full path:

```
chmod o-r /home/sue/budget.txt
```

If you're using the `chmod` command against files other than those you own yourself, you'll need to use `sudo`.

But either way, you probably get the idea. With this example, we're removing the `r` bit from *other* (`o-r`). If we wanted to add this bit instead, we would simply use `+` instead of `-`. Here are some additional examples of `chmod` in action:

- `chmod u+rw <filename>`: The object gets `rw` added to the user column
- `chmod g+r <filename>`: The owning group is given read access
- `chmod o-rw <filename>`: *Other* is stripped of the `rw` bits

In addition, you can also use octal point values to manage and modify permissions. This is actually the most common method of altering permissions. I like to think of this as a scoring system. That's not what it is, but it makes it a lot easier to understand to think of each type of access as having its own value. Basically, each of the permission bits (`r`, `w`, and `x`) has its own octal equivalent, as follows:

- Read: 4
- Write: 2
- Execute: 1

With this style, there are only a few possibilities for numbers you can achieve when combining these octal values (each can only be used once). Therefore, we can get 0, 1, 2, 3, 4, 5, 6, and 7 by adding (or not adding) these numbers in different combinations. Some of them you'll almost never see, such as an object having write access but not read. For the most part, you'll see 0, 4, 5, 6, and 7 used with `chmod` most often. For example, if we add Read and Write, we get 6. If we add Read and Execute, we get 5. If we add all three, we get 7. If we add no permissions, we get 0. We repeat this for each column (User, Group, and Other) to come up with a string of three numbers. Here are some examples:

- `600`: User has read and write (4+2). No other permissions are set.
This is the same as `-rw-----`.
- `740`: User has read, write, and execute. Group has read. Other has nothing.

This is the same as `-rwxr-----`.

- **770:** Both User and Group have full access (read, write, and execute). Other has nothing.

This is the same as `-rwxrwx---`.

- **777:** Everyone has everything.

This is the same as `-rwxrwxrwx`.

Going back to `chmod`, we can use this numbering system in practice:

- `chmod 600 filename.txt`
- `chmod 740 filename.txt`
- `chmod 770 filename.txt`

Hopefully you get the idea. If you wanted to change the permissions of a directory, the `-R` option may be helpful to you. This makes the changes recursive, meaning that you'll not only make the changes to the directory but also to all files and directories underneath it in one shot:

```
chmod 770 -R mydir
```

While using `-R` with `chmod` can save you some time, it can also cause trouble if you have a mix of directories and files underneath the directory you're changing permissions on. The previous example gives permissions `770` to `mydir` and all of its contents. If there are files inside, they are now given executable permissions to the user and group, since `7` includes the execute bit (value of `1`). This may not be what you want. We can use the `find` command to differentiate these. While `find` is out of the scope of this chapter, it should be relatively simple to see what the following commands are doing and how they may be useful:

```
find /path/to/dir/ -type f -exec chmod 644 {} \;
find /path/to/dir/ -type d -exec chmod 755 {} \;
```

Basically, in the first example, the `find` command is locating all files (`-type f`) in `/path/to/dir/` and everything it finds, it executes `chmod 644` against. The second example is locating all directories in this same path and changing them all to permission `755`. The `find` command isn't covered in detail here because it easily deserves a chapter of its own, but I'm including it here because hopefully these examples are useful and will be handy for you to include in your own list of useful commands.

Changing the ownership of objects

Finally, we'll need to know how to change the ownership of files and directories. It's often the case that a particular user needs to gain access to an object, or perhaps we need to change the owning group as well. We can change user and group ownership of a file or directory with the `chown` command. As an example, if we wanted to change the owner of a file to `sue`, we could do the following:

```
sudo chown sue myfile.txt
```

In the case of a directory, we can also use the `-R` flag to change the ownership of the directory itself, as well as all the files and directories it may contain:

```
sudo chown -R sue mydir
```

If we would like to change the group assignment to the object, we would follow the following syntax:

```
sudo chown sue:sales myfile.txt
```

Notice the colon separating the user and the group. With that command, we established that we would like the `sue` user and the `sales` group to own this resource. Again, we could use `-R` if the object were a directory and we wanted to make the changes recursive.

Another command worth knowing is the `chgrp` command, which allows you to directly change the group ownership of a file. To use it, you can execute the `chgrp` command along with the group you'd like to own the file, followed by the filename. For example, our previous `chown` command can be simplified to the following, since we were only modifying the group assignment of that file:

```
sudo chgrp sales myfile.txt
```

Just like the `chown` command, we can use the `-R` option with `chgrp` to make our changes recursively, in the case of a directory.

Well, there you have it. You should now be able to manage permissions of the files and directories on your server. If you haven't worked through permissions on a Linux system before, it may take a few tries before you get the hang of it. The best thing for you to do is to practice. Create some files and directories (as well as users) and manage their permissions. Try to remove a user's access to a resource and then try to access that resource as that user anyway and see what errors you get. Fix those errors and work through more examples. With practice, you should be able to get a handle on this very quickly.

Summary

In Linux administration and related fields, managing users and permissions is something you'll find yourself doing quite a bit. New users will join your organization, while others will leave, so this is something that will become ingrained in your mental toolset. Even if you're the only person using your servers, you'll find yourself managing permissions for applications as well, given the fact that processes cannot function if they don't have access to their required resources.

In this chapter, we took a lengthy dive into managing users, groups, and permissions. We worked through creating and removing users, assigning permissions, and managing administrative access with `sudo`. Practice these concepts on your server. When you get the hang of it, I'll see you in our next chapter, where we'll discuss all things related to package management. It's going to be epic.

Relevant videos

- Linux Crash Course – Managing Users (LearnLinuxTV): <https://linux.video/lcc-users>
- Linux Crash Course – Understanding File & Directory Permissions (LearnLinuxTV): <https://linux.video/lcc-perm>

- Linux Crash Course – usermod (LearnLinuxTV): <https://linux.video/lcc-usermod>
- Linux Crash Course – sudo (LearnLinuxTV): <https://linux.video/lcc-sudo>
- Linux Crash Course – User Account & Password Expiration (LearnLinuxTV): <https://linux.video/lcc-userexp>
- Linux Crash Course – Managing Groups (LearnLinuxTV): <https://linux.video/lcc-groups>

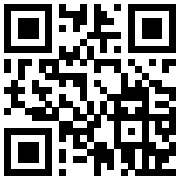
Further reading

- File permissions (Ubuntu community wiki): <https://learnlinux.link/ubuntu-perms>
- User management (Ubuntu documentation): <https://learnlinux.link/sec-users>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



3

Managing Software Packages

Now that you have a server installation set up, and you know how to manage users on it, it's time to cover the management of software. The Ubuntu platform has a huge range of software available, featuring packages for everything from server administration to games. In fact, as of the time I'm writing this chapter, there are over 60,000 packages in Ubuntu's repositories. That's a lot of software packages, and in this chapter, we'll take a look at how to manage them. We'll cover how to install, remove, and update packages, as well as the use of related tools.

As we go through these concepts, we will cover:

- Understanding Linux package management
- Understanding the differences between Debian and Snap packages
- Installing and removing software
- Searching for packages
- Managing package repositories
- Backing up and restoring Debian packages
- Cleaning up orphaned APT packages
- Taking advantage of hardware enablement updates

To get started, let's build an understanding of how software packages are distributed in Ubuntu, and the basic principles of the concept of package management.

Understanding Linux package management

Nowadays, *app stores* are all the rage on most platforms; typically, you'll have one central location from which to retrieve applications, allowing you to install them on your device. Even phones and tablets utilize a central software repository in which software is curated and made available. The Android platform has Google Play, Apple has its App Store, and so on. For those that have used Linux for a while, this concept isn't new. The concept of software repositories is similar to that of app stores and has been around within the Linux community since long before cellular phones even had color screens.

Linux has had package management since the '90s, initially popularized by **Debian** and then **Red Hat**. Software repositories are generally made available in the form of **mirrors**, to which your server subscribes. Mirrors are available across a multitude of geographic areas, so, typically, your installation of Ubuntu Server would subscribe to the mirror closest to you. These mirrors are populated with software packages that you'll be able to install. Many packages depend on other packages, so various tools on the Linux platform exist to automatically handle these dependencies for you. Not all distributions of Linux feature package management and dependency resolution, but Ubuntu certainly does, benefiting from the groundwork already built by Debian.

Packages contained within these mirrors are constantly changing. Traditionally, an individual known as a **package maintainer** is responsible for one or more packages and ships new versions to the repositories for approval and, eventually, distribution to mirrors. Specific to Ubuntu's repositories, a group of developers, rather than just a single maintainer, is responsible for maintaining packages. Most of the time, the new version of a package is provided in order to patch a security vulnerability, but otherwise it contains no new features. With the majority of Ubuntu's packages being open source, anyone is able to look at the source code, find problems, and report issues. When vulnerabilities are found, the responsible team will review the claim and then release an updated version to correct it. This process happens very quickly, and I've seen severe vulnerabilities patched even on the same day that they were reported in some cases. Ubuntu developers are definitely on top of their game in terms of taking care of security issues.

New versions of packages are also sometimes feature updates, which are updates released to introduce new features not necessarily tied to a security vulnerability. This could be a new version of a desktop application such as Firefox or a server package such as MySQL. Most of the time, though, new versions of packages that are vastly different are held for the next Ubuntu release. The reason for this is that too much change can cause your server to not be as stable, and experience application or even entire OS-level crashes. Instead, known working and stable packages are preferred; however, given the fact that Ubuntu releases every six months, you don't have to wait very long.

Specifically, feature updates in an Ubuntu release must go through an approval process before being made available in the default repositories, and these feature updates are specifically referred to as **Stable Release Updates (SRUs)**. There's an entire process around how these updates are approved, but the general takeaway is that there should be a good reason to implement a major version change in a stable, long-term Ubuntu release.

As a server administrator, you'll often need to make a choice between security and feature updates. Security updates are the most important of all and allow you to patch your servers in response to security vulnerabilities. Sometimes, feature updates become required in your organization because it's decided that new features may benefit you or may become required for current objectives. In this chapter, we won't focus on installing security updates (we'll take care of that in *Chapter 21, Securing Your Server*), but it's important to understand the reasons new packages are made available to you.

Package management is typically very convenient in Ubuntu, with security updates and bug fixes coming regularly. With just one command (which we'll get to shortly), you can install a package along with all of its dependencies. Having performed manual dependency resolution myself, I can tell you first-hand that having dependencies handled automatically is a wonderful thing.

The main benefit of how packages are maintained on a Linux server is that you generally don't have to search the internet for packages to download, as Ubuntu's repositories contain most of the ones you'll ever need. As we continue through this chapter, you'll come to know everything you need in order to manage this software.

Understanding the differences between Debian and Snap packages

Now, before we get into the ins and outs of managing packages, there are actually two completely different types of packages available to you, and you should understand the differences between them. As of the time of writing, we're at a kind of crossroads regarding the way in which software is managed in Linux.

Traditionally, each distribution has its own package format and its own utilities to manage them. Ubuntu utilizes **Debian packages** (with package names ending in `.deb`) as the main package format, which Ubuntu inherits from the Debian distribution (Ubuntu is forked from Debian, which means that it uses Debian as a foundation). Ubuntu and Debian utilize the `apt` and `dpkg` commands to manage packages. On the other hand, distributions such as CentOS and Red Hat use **RPM packages** for their distributions, and the `dnf` command to manage them. There are other distributions and package formats as well, but for this book, we'll stick with what's available in Ubuntu. First, let's discuss Debian packages.

Debian packages

Debian packages have been the main type of package in Ubuntu for the entire existence of the distribution so far. When you search online for how to install something in Ubuntu, chances are, you're going to be installing a Debian package. These packages are known as Debian packages because Ubuntu is built from Debian sources and utilizes the same commands in order to install these packages. So even though Ubuntu is not Debian (Debian is a completely different distribution), they both use the same package format primarily.

The naming may be confusing for newcomers, because if Ubuntu is considered a different distribution than Debian, then why refer to its packages as "Debian" packages? Debian packages have a filename that ends in `.deb`, and this package format originated in Debian. Ubuntu didn't develop its own package type; it uses the same package format as Debian. Therefore, whether we are installing packages in Debian or Ubuntu using a command such as `apt`, Debian packages are the type of packages used for both.

If you've worked with Ubuntu before reading this book, then chances are, you've already used the `apt` series of commands to carry out some package management. Debian packages are great because when paired with the `apt` command, they are easy to use and handle dependency resolution for you. However, they present some challenges and major drawbacks.

First, the majority of the distribution is made up of Debian packages. This means that the Linux kernel, system packages, libraries, and security updates are all Debian packages that are installed when you install Ubuntu Server. When you install security updates, Debian packages are installed. The reason this may be a problem is that other software you'll be installing, such as Apache, MariaDB, and so on, are also Debian packages, and may conflict with system packages when one package requires a prerequisite package that conflicts with another.

This can lead to a situation where you can't install packages at all. Package maintainers are generally good at avoiding this scenario, so conflicts aren't incredibly common nowadays.

However, with Debian packages, if a system library gets corrupted, literally every piece of software that depends on it will fail. Ubuntu developers pay a great deal of attention to this, so you shouldn't run into issues. But the truth is, this is a lot of work for the maintainers of Ubuntu to deal with.

Another concern with Debian packages is software availability. When a new major version of a package is released, it generally will not be offered to you until the next release of the distribution. This means that if you want a version of PHP, Apache, or some other piece of software that's newer than what your current release of Ubuntu features, you generally will not have it offered to you. Instead, you typically wait until the next release of the entire distribution. There are some exceptions to this, such as Firefox in the desktop version of Ubuntu. As mentioned before, new major package versions are exceptions to the rule and come from the SRU approval process. While having tried-and-true software available that has been extensively tested offers better stability, sometimes you may require a newer version of particular software than what is available, which may lead you to consider alternative sources. After all, you don't have to install a new version of Windows or macOS just to install newer applications!

Universal packages are a new concept for Linux and are intended to be a single package format that multiple distributions recognize. The idea is that a developer only has to compile one package (instead of a separate package for each distribution) and users would only need to download a single package regardless of their chosen Linux flavor. The next section will discuss a type of universal package called Snaps.

Snap packages

As it stands today, developers would need to create multiple package types to support Linux. Perhaps they'll create Debian packages for Debian itself and Ubuntu, and then RPM packages for CentOS, Red Hat Enterprise Linux, and SuSE. And while you may think that having to create two package types isn't all that bad, consider that each of the RPM-based distributions needs its own specific RPM package, and you can see how it might be tedious to a developer to have to create five or more different packages for any one release of their software.

As a result of this, there's a push to adopt a single package format that each distribution can install that is independent of the system package type. This concept is known as **universal packages**, and the idea is to have a standard package type that can be installed on any Linux distribution. As a result, developers would only have to create one package to have their application work on all the popular distros.



The Linux community commonly uses the word *distro* as a contraction of the term *distribution*.

Another benefit of universal packages is that they have all of their dependencies built in, so conflicts are less likely to occur; everything the application needs would be contained in one single package. This is great because the likelihood of you running into package conflicts with universal packages is next to non-existent.

As with all things in the IT industry, we can't collectively be satisfied and decide on the proper technology. As such, there is debate among the community regarding which one of the multiple types of universal packages is the most suitable. I won't get into the political debates in this book, as each has its strengths and weaknesses. But even easier for us in regards to Ubuntu Server, only one of them is ideal for server installations anyway. The competing technologies for universal packages include Flatpak, AppImage, and Snap packages.

Canonical, the makers of Ubuntu, understand the pain points that developers and users experience, and have been making a great effort to change how packages are managed. The type of universal package they've developed to address these concerns is known as the **Snap package**. Like all universal packages, Snap packages (or more simply, *Snaps*) have no impact on the underlying Debian packages at all and are completely independent, thus removing the possibility of conflicts with your system packages. This allows you to have a newer version of an application installed than what would otherwise be made available. Since Snaps are installed separately and independently from the underlying Debian packages, there's no reason to withhold them. Snap packages are better in just about every way and are a great concept. The only downside might be that they are larger packages, since they include not only the application itself but also all the libraries they require in one single package. However, they're really not that large and shouldn't cause an issue with disk space. These packages are no bigger than a typical application on macOS or Windows.

So, which should you choose? It really just depends on your use case. Each of the universal package types is good at some things but has downsides. Flatpaks and AppImages are also great technologies, but where they fall behind for our use case is that they don't support server (non-**Graphical User Interface**, or non-GUI) applications as well. This means that these package types are best for installing applications you'd normally find on a desktop Ubuntu installation, such as music players, browsers, graphical text editors, and so on. Snap packages target *both* GUI and non-GUI apps. Since server installations normally don't contain a GUI at all, this makes our choice for us. It could be the case that Flatpaks and AppImages start supporting non-GUI apps better in the future, but for now, the choice of universal package format is simple.

As it stands today, though, the majority of the packages we'll be installing are going to be Debian packages, as Snap packages are a bit slow to catch on. They're making steady progress, but it remains to be seen how well the industry will embrace them. For now, it's probably best to evaluate Snap packages when they're available and to choose the most appropriate package type based on availability, security, version, and support.

With the differences between Snaps and Debian packages out of the way, let's work through some examples of how to actually manage the software on our server. We'll look at commands to search for available packages, and then we'll install them and remove them.

Installing and removing software

Before we begin, we will want to research a bit regarding the application we want to install. In Ubuntu, there are multiple ways of installing software, so the best way to find out how to get started is by simply checking the documentation on the website for the application we want to install. Typically, a Google search will do (just make sure you check the domain and are on the correct site). Most software will have installation instructions for various platforms, including Ubuntu. Most of the time, it will just tell you to download the Debian package format via the `apt install` command. Other times, the software may have a Snap available, or even a PPA repository (we'll discuss PPA repositories later on in this chapter). Let's start our package management journey by taking a look at the `apt` commands that are used to install Debian packages.

Managing Debian packages with apt

APT, or **Advanced Package Tool**, is a suite of tools that allows us to install, remove, and update Debian packages. There are various sub-commands that make up this suite, which we'll go over now. The most popular variation of the `apt` command is `apt install`. If you've ever read instructions for how to do something in Ubuntu, chances are you've already run that command to install a package. And that's exactly what it does: it allows you to install packages for Ubuntu over the command line. For example, the following command will install the `openssh-server` package:

```
sudo apt install openssh-server
```

You can also install multiple packages at a time by separating each with a space, instead of installing each package one at a time. The following example will install three different packages:

```
sudo apt install <package1> <package2> <package3>
```

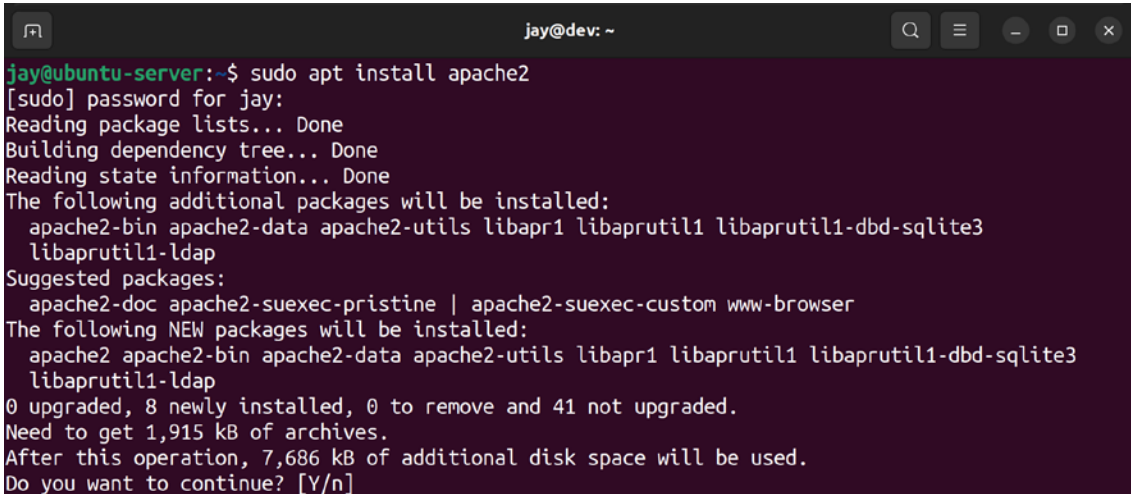
In some books, blogs, and articles, you may see longer versions of `apt` commands, such as `apt-get install` instead of just `apt install`. Being able to shorten commands such as `apt-get install` to `apt install` is a relatively new feature of `apt` in Debian and Ubuntu. Both methods are perfectly valid, but simplifying APT commands down to just `apt` is preferred going forward.

So, what actually happens when you install a package with `apt`? If you've run through the process before, you're probably accustomed to it already. But, typically, the process begins with `apt` calculating dependencies. The majority of packages require other packages to function, so `apt` will check to ensure that the package you're requesting is available and that its dependencies are available as well.

First, you'll see a summary of the changes that apt wants to make to your server. In the case of installing the apache2 package on an unconfigured Ubuntu Server, I enter the following command:

```
sudo apt install apache2
```

I see the following output on my system when it starts to install:



```
jay@dev: ~  
jay@ubuntu-server:~$ sudo apt install apache2  
[sudo] password for jay:  
Reading package lists... Done  
Building dependency tree... Done  
Reading state information... Done  
The following additional packages will be installed:  
  apache2-bin apache2-data apache2-utils libapr1 libaprutil1 libaprutil1-dbd-sqlite3  
  libaprutil1-ldap  
Suggested packages:  
  apache2-doc apache2-suexec-pristine | apache2-suexec-custom www-browser  
The following NEW packages will be installed:  
  apache2 apache2-bin apache2-data apache2-utils libapr1 libaprutil1 libaprutil1-dbd-sqlite3  
  libaprutil1-ldap  
0 upgraded, 8 newly installed, 0 to remove and 41 not upgraded.  
Need to get 1,915 kB of archives.  
After this operation, 7,686 kB of additional disk space will be used.  
Do you want to continue? [Y/n]
```

Figure 3.1: Installing Apache on an example server

Even though I only asked for apache2, apt informs me that it also needs to install apache2-bin, apache2-data, apache2-utils, and libapr1 (and others) in order to satisfy the dependencies for the apache2 package. apt also suggests that I install apache2-doc, apache2-suexec-pristine, and a few others, though they are optional and are not required. You can install the suggested packages by adding the `--install-suggests` option to the `apt install` command, but that isn't always a good idea as it may install a large number of packages that you may not need. You can, of course, cherry-pick the suggested packages individually by using the `apt` command to install any one or more of them manually.

Most of the time, though, you probably won't want to do this; it's usually better to keep your installed packages to a lean minimum and install only the packages you need. As we'll discuss in *Chapter 21, Securing Your Server*, the fewer packages you install, the smaller the attack surface of your server.

While installing or updating packages via `apt`, you might see a message come up that asks you whether or not you're comfortable with having background services restarted as part of the process:

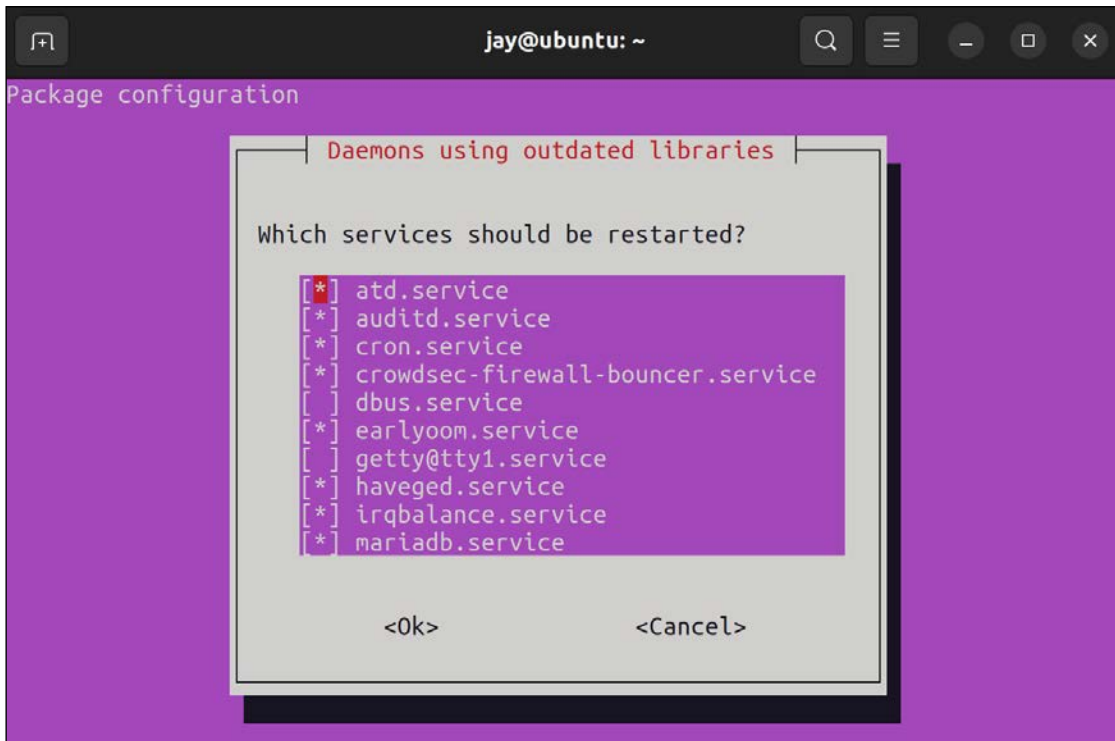


Figure 3.2: Service restart prompt during package installation/upgrades

We'll cover concepts around process management in *Chapter 7, Controlling and Managing Processes*, so it's not something you should focus on right now. For a quick description in order to hold you over until then, there's services running in the background on all Linux servers, and each one provides a particular function. Restarting a service will ensure it restarts with all the latest patches and tweaks applied, but any user connected to one of the services you restart will get disconnected. For now, you can simply press Enter to accept the defaults anytime you see this message.

Another option that is common with installing packages via `apt` is the `-y` option, which assumes *yes* to the confirmation prompt where you choose if you want to continue or not. For example, my previous output included the line `Do you want to continue? [Y/n]`. If we had used `-y`, the command would have proceeded to install the package without any confirmation. This can be useful for administrators in a hurry, though I personally don't see the need for this unless you are scripting your package installations. In fact, it can sometimes be a bad idea, because by assuming *yes*, you may be confirming something you'd later regret.

Another neat default in Ubuntu Server is that it automatically configures most packages to make their daemons start up and also be enabled so that they start with each boot. Using the earlier example of `apache2`, the `apache2` service will start and the application will automatically start running as soon as you install the package.

This may seem like a good idea for the sake of convenience, but not everyone prefers this automation. As I've mentioned, the more packages installed on your server, the higher the attack surface, but running services (also known as daemons, or units) are each a method of entry for miscreants should there be a security vulnerability. Therefore, some distributions don't enable and start daemons automatically when you install packages. The way I see it, though, you should only install packages you actually intend to use, so it stands to reason that if you go through the trouble of manually installing a package such as `apache2`, you probably want to start using it.

When you install a package with the `apt` command, it searches its local database for the package you named. If it doesn't find it, it will throw an error. Sometimes, this error may be because the package isn't available or perhaps the version that `apt` wants to install no longer exists. Ubuntu's repositories move very quickly; new versions of packages are added almost daily. When a new version of a package is added, its older equivalent may be removed. For this reason, it's recommended that you update your package sources from time to time. Doing so is easy, using the following command:

```
sudo apt update
```

This command doesn't actually update any packages; it merely checks in with your local mirror to see if any packages have been added or removed and updates your local index. This command is useful because installations of packages can fail if your sources aren't up to date. In most cases, the symptom will be that `apt` errors out of the process when it can't find a package it's looking for, something that would've been available normally.

Removing packages is also very easy and follows a very similar syntax; you would only need to replace the `install` keyword with `remove`:

```
sudo apt remove <package>
```

And, just like with the `install` option, you can remove multiple packages at the same time. The following example will remove three packages:

```
sudo apt remove <package1> <package2> <package3>
```

If you'd like to not only remove a package but also wipe out its configuration, you can use the `--purge` option:

```
sudo apt remove --purge <package>
```

This will not only remove the package but wipe out its configuration directory (applications typically store their configuration files in a sub-directory of `/etc`).

So, that concludes the basics of managing Debian packages with `apt`. Now, let's move on to managing Snaps.

Managing Snap packages with `snap`

To manage Snap packages, we use the `snap` command. The `snap` command features several options we can use to search for, install, and remove Snap packages from our server or workstation.

To begin, we can use the `snap find` command along with a keyword to display a list of Snap packages available to us that match that keyword:

```
snap find <keyword>
```

Basically, you just simply type the `snap find` command along with a search term to look for. One example could be the `nmap` application, which is a useful tool to have if we're managing a network:

```
snap find nmap
```

In the case of `nmap`, this utility is available in Ubuntu's default repositories, so you don't need to use the Snap package to install it. Typically, though, the Snap version will be newer and have more features than what is available in the APT repositories. If we wish to install the Snap version, we can use the following command to do so:

```
sudo snap install nmap
```

With the `install` option, we need to use `sudo` since the act of installing a package makes changes on the server. For a simple `find`, we can omit `sudo`.

Now that we have `nmap` installed, we can check the location of the `nmap` binary with the `which` command. We can use the `which` command to find the location of the binary file for commands that are available—it will show the path to the binary if the appropriate package is installed. If the command is not available, the `which` command will show no output. So, if we run the following command, we should now see the path of that binary printed to the screen:

```
which nmap
```

This will return the following output, showing that the Snap version of `nmap` is run from a special place:

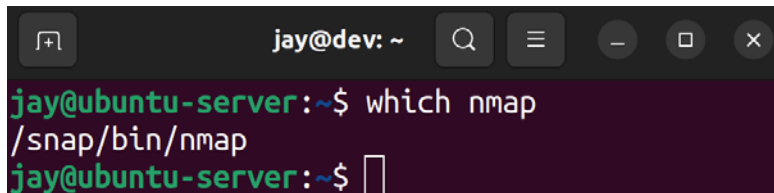
A terminal window screenshot showing the command 'which nmap' being executed. The prompt is 'jay@ubuntu-server:~\$'. The output is '/snap/bin/nmap'. The prompt then returns to 'jay@ubuntu-server:~\$' with a cursor. The window title is 'jay@dev: ~' and it has standard Ubuntu window controls.

Figure 3.3: Checking the location of the `nmap` binary

Now, when we run `nmap`, we're actually running it from `/snap/bin/nmap`. If we were to install `nmap` via `apt`, it would run from `/usr/bin/nmap` instead. If we also have the `nmap` utility installed from Ubuntu's APT repositories, then we can run either one at any time by calling out the full path to the binary we want to run, since Snap packages are independent of the APT packages. For example, if we had `nmap` installed via both `snap` and `apt`, we could run Ubuntu's version by running `/usr/bin/nmap` and the Snap version by running `/snap/bin/nmap`.

Removing an installed Snap package is easy. We simply use the `remove` option:

```
sudo snap remove nmap
```

If we issue that command, then `nmap` (or whichever Snap package we designate) is removed from the system.

To update a package, we use the `refresh` option along with the name of a package to update:

```
sudo snap refresh <package>
```

With that command, the package will be updated to the newest version available. Going even further, we can attempt to update every Snap on our server with the same command (without specifying a package):

```
sudo snap refresh
```

As you can see, managing Snap packages is fairly straightforward. Using the `snap` suite of commands, we can install, update, or remove packages from our server or workstation. The `snap find` command allows us to find new Snap packages to install. Perhaps as the technology matures, we may find ourselves installing more Snap packages than Debian packages, but that remains to be seen. For now, it's a nice benefit to have two options to consider when installing new software.

In addition to being able to install packages, there are some additional tips and considerations around searching for packages. After all, you can't install a package if you don't know what's available. We'll explore searching for packages in the next section.

Searching for packages

Unfortunately, the naming conventions used for packages in Ubuntu Server aren't always obvious. Worse, package names are often very different from one distribution to another even for the same piece of software. While this book and other tutorials online will outline the exact steps needed to install software, if you're ever on your own, it really doesn't help much if you don't know the name of the package you want to install. In this section, I'll try to take some of the mystery out of searching for packages.

In the previous section, we went over searching for Snap packages, so I won't repeat that here. The APT suite of utilities also has a means of searching for packages as well, which is the `apt search` command. We can use the following command to search for packages, by providing a keyword:

```
apt search <search term>
```

The output from this command will show a list of packages that match your search criteria, with their names and descriptions. If, for example, you wanted to install the PHP plugin for Apache and you didn't already know the name of the associated package, the following would narrow it down:

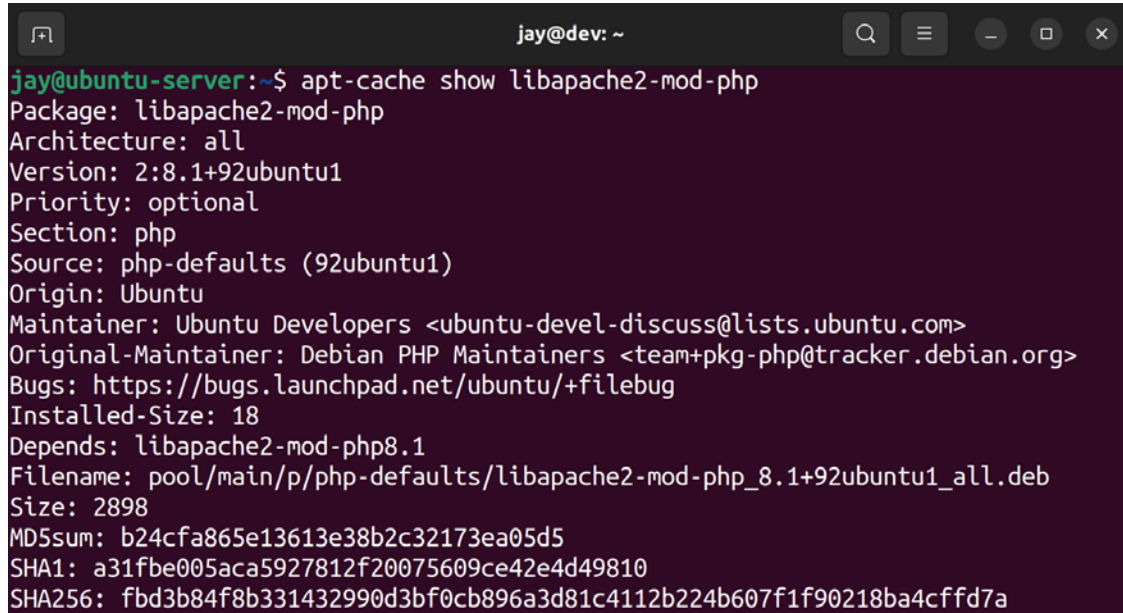
```
apt search apache php
```

In the output, we will get a list of more than a handful of packages, but we can deduce from the package descriptions in the output that `libapache2-mod-php` is most likely the one we want. We can proceed to install it using `apt`, as we would normally do.

If we're not sure whether or not this is truly the package we want, we can view more information regarding this (or any other) package with the `apt-cache show` command:

```
apt-cache show libapache2-mod-php
```

The output of this command is illustrated in the following screenshot:

A terminal window titled 'jay@dev: ~' with search, menu, and window control icons in the title bar. The terminal shows the command 'apt-cache show libapache2-mod-php' and its output. The output lists package details for libapache2-mod-php, including architecture, version, priority, section, source, origin, maintainer, original maintainer, bugs, installed size, dependencies, filename, and checksums.

```
jay@ubuntu-server:~$ apt-cache show libapache2-mod-php
Package: libapache2-mod-php
Architecture: all
Version: 2:8.1+92ubuntu1
Priority: optional
Section: php
Source: php-defaults (92ubuntu1)
Origin: Ubuntu
Maintainer: Ubuntu Developers <ubuntu-devel-discuss@lists.ubuntu.com>
Original-Maintainer: Debian PHP Maintainers <team+pkg-php@tracker.debian.org>
Bugs: https://bugs.launchpad.net/ubuntu/+filebug
Installed-Size: 18
Depends: libapache2-mod-php8.1
Filename: pool/main/p/php-defaults/libapache2-mod-php_8.1+92ubuntu1_all.deb
Size: 2898
MD5sum: b24cfa865e13613e38b2c32173ea05d5
SHA1: a31f8e005aca5927812f20075609ce42e4d49810
SHA256: fbd3b84f8b331432990d3bf0cb896a3d81c4112b224b607f1f90218ba4cffd7a
```

Figure 3.4: Showing the info of a Debian package

With this command, we can see additional details regarding the package we're considering installing. In this case, we learn that the `libapache2-mod-php` package also depends on PHP itself, so that means if we install this package, we'll get the PHP plugin as well as PHP itself.

Another method of searching for a package (if you have a web browser available) is to connect to the *Ubuntu Packages Search* page at <https://packages.ubuntu.com/>, where you can navigate through the packages from their database for any currently supported version of Ubuntu. You won't always have access to a web browser while working on your servers, but, when you do, this is a very useful way to search through packages, view their dependencies, descriptions, and more.

Using the `apt search` command, as well as the `snap find` command, should get you quite far in the process of determining the name of the packages you want to install. Package management skills come over time, so don't expect to automatically know which packages to install right away. When in doubt, just perform a Google search, research the documentation of the software you want to run, and learn how to install it in Ubuntu. Typically, the instructions will lead you to the correct commands to use. The examples we'll go over during the course of this book will guide you through the most common use cases for Ubuntu Server.

At this point, we should have a solid understanding of the different types of packages available and how to manage them. However, it's sometimes the case that we need to run software on our server for which there is no package available within the standard repositories. Therefore, in the next section, we'll learn how to add additional repositories from which to install software.

Managing package repositories

Often, the repositories that come pre-installed with Ubuntu will suffice for the majority of the Debian packages you'll install via APT. Every now and then, though, you may need to install an additional repository in order to take advantage of software not normally provided by Ubuntu, or versions of packages newer than what you would normally have available. Adding additional repositories allows you to subscribe to additional sources of software and install packages from them in the same way as you would from any other source.

Adding additional repositories should be considered a last resort, however. When you install an additional repository, you're effectively trusting the author of that repository with your organization's server. Although I haven't ever seen this happen first-hand, it's theoretically possible for authors of software to include back doors or malware in software packages (intentionally or unintentionally), and then make them available for others via a software repository. Therefore, you should only add repositories from sources that you have reason to trust.

In addition, it sometimes happens that a maintainer of a repository simply gives up on it and disappears. This I have seen happen first-hand. In this situation, the repository may go offline (which would show errors during apt transactions, indicating that it's not able to connect to the repository), or worse, the repository stays online, but security updates are never made available, causing your server to become vulnerable to attack. Sometimes, you just don't have a way around it. You need a specific application and Ubuntu doesn't offer it by default. Your only option may be to compile an application from source or add a repository. The decision is yours, but just keep security in mind whenever possible. When in doubt, avoid adding a repository unless it's the only way to obtain what you're looking for. If you do add an additional repository and you haven't seen packages from it updated in a decent amount of time, consider that a red flag and investigate whether or not the repository is abandoned. Repositories that are abandoned should be removed, and an alternative used in its place.

Adding additional repositories

Software repositories are essentially URLs in a text file, stored in one of two places. The main Ubuntu repository list is stored in `/etc/apt/sources.list`. Inside that file, you'll find a multitude of repositories for Ubuntu's package manager to pull packages from. In addition, files with an extension of `.list` are read from the `/etc/apt/sources.list.d/` directory and are also used whenever you use apt. I'll demonstrate both methods.

A typical repository line in either of these two files will look similar to the following:

```
deb http://us.archive.ubuntu.com/ubuntu/ jammy main restricted
```

The first section of each line will be either `deb` or `deb-src`, which references whether the `apt` command will find binary packages (`deb`) or source packages (`deb-src`) there. Next, we have the actual URL that `apt` will use in order to reach the repository. In the third section, we have the codename of the release; in this case, it's `jammy` (which refers to the codename for Ubuntu 22.04, `Jammy Jellyfish`).

If you weren't already aware, the codename for each Ubuntu release is based on an animal. Often, the animal chosen for a release will be rare. For example, the previous LTS release featured the `fossa`, which is an animal from Madagascar that somewhat resembles a cat but with curved ears. This time around, the chosen animal is the `jellyfish`, which I'm sure the majority of my readers are already familiar with.

Continuing, the fourth section of each repository line refers to the `Component`, which references whether or not the repository contains software that is free and open source, and is supported officially by Canonical (the company that oversees Ubuntu's development). The component can be `main`, `restricted`, `universe`, or `multiverse`. Repositories with a `main` component include officially supported software. This generally means that the software packages have source code available, so Ubuntu developers are able to fix bugs. Software marked `restricted` is still supported but may have a questionable license. `universe` packages are supported by the community, not Canonical themselves. Finally, `multiverse` packages contain software that is neither free nor supported, which you would be using at your own risk.

As you can see from looking at the `/etc/apt/sources.list` file on your server, it's possible for a repository line to feature software from more than one component. Each repository URL may include packages from several components, and the way you differentiate them is to only subscribe to the components you need for that repository. In our previous example, the repository line included both `main` and `restricted` components. This means that, for that particular example, the `apt` utility will index both free (`main`) and non-free (`restricted`) packages from that repository.

You can add new repositories to the `/etc/apt/sources.list` file (and it will function just fine), but that's not typically the preferred method. Instead, as I mentioned earlier, `apt` will scan the `/etc/apt/sources.list.d/` directory for text files ending with the `.list` extension. These text files are formatted the same as the `/etc/apt/sources.list` file in the sense that you include one additional repository per line, but this method allows you to add a new repository by simply creating a file for it, and you can remove the repository by simply deleting that file.

This is safer than editing the `/etc/apt/sources.list` file directly, since there's always a chance you can make a typo and disrupt your ability to download packages from even the official repositories.

In addition, you may need to install a **GNU Privacy Guard (GnuPG)** key for a new repository, but this process differs from one application to another. Typically, the documentation will outline the entire process. This key basically protects you in that it makes sure that you're installing signed packages. Not all developers protect their applications this way, but it's definitely a good thing to do.

Once you have the repository (and possibly the key) installed on your server, you'll need to run the following command to update your package index:

```
sudo apt update
```


As mentioned earlier in this chapter, this command updates your local cache as to which packages are available on the remote server. APT is only aware of packages that are in its database, so you will need to sync this with that command before you'll be able to actually install the software contained within the repository.

Adding Personal Package Archives

On the Ubuntu platform, there also exists another type of repository, known as a **Personal Package Archive (PPA)**. PPAs are essentially another form of APT repository, and you'll even interact with their packages with the `apt` command, as you would normally. PPAs are usually very small repositories, often including a single application that serves a single purpose. Think of PPAs as *mini-repositories*. A PPA is common in situations where a vendor doesn't make their software available with their own repository and may only make their application available in the form of source code you would need to manually download, compile, and install. With the PPA platform, anyone can compile a package from source and easily make it available for others to download.

PPAs suffer from the same security concerns as regular repositories (you need to trust the vendor and so on), but are a bit worse considering that the software typically isn't audited at all. In addition, if the PPA was to ever go down, you'd stop getting security updates for the application you install from it. Only use PPAs when you absolutely need to.

There is one use case for PPAs that may be compelling, specifically for a server platform that standard repositories aren't able to handle very well, and that is software versioning. As I mentioned earlier, a major server component such as PHP or MySQL may be locked to a specific major version with each Ubuntu Server release. What do you do if you need to use Ubuntu Server, but the application you need to run is not available in the version your organization requires? In the past, you would literally need to choose between the distribution and the package, with some organizations even using a different distribution of Linux just to satisfy the need to have a specific application at a specific version. You can always compile the application from source (assuming its source code is available), but that can cause additional headaches in the sense that you'd be responsible for compiling new security patches yourself whenever they're made available. PPAs potentially give you access to applications not normally available in the default repositories, and/or access to newer versions of packages than what is normally provided. This gives you, the server administrator, the ability to choose the approach that is best for your goal.

PPAs are generally added to your server with the `apt-add-repository` command. The syntax generally uses the `apt-add-repository` command, with a colon, followed by a username, and then the PPA name. The following command is a hypothetical example:

```
sudo apt-add-repository ppa:username/myawesomesoftware-1.0
```

To begin the process, you would start your search by visiting Ubuntu's PPA website, which is available at <https://launchpad.net/ubuntu/+ppas>. There, you can search among the available PPAs.

Before adding a PPA to your server, it's best to first research whether or not it's being maintained well. For example, if the PPA hasn't had any updated packages in a very long time, that's cause for concern—security fixes are fairly common with most packages. If a package isn't being regularly updated and has therefore gone “stale,” it may be best to avoid it as it may cause more harm than good.

Once you find a PPA you would like to add to your server, you can add it simply by finding the name of the PPA and then adding it to your server with the `apt-add-repository` command.

You should take a look at the page for the PPA, though, in case there are different instructions. For the most part, the `apt-add-repository` command should work fine for you. Each PPA typically has a set of instructions attached, so there shouldn't be any guesswork required here.

So, what exactly does the `apt-add-repository` command do? Honestly, it's not all that amazing. When you install a PPA, it's essentially automating the process of adding a repository file to your `/etc/apt/sources.list.d` directory and installing its key. Therefore, you can uninstall a PPA by simply deleting its file.

PPAs are a very useful feature if harnessed with care. PPAs offer Ubuntu a flexible way of adding additional software that wouldn't normally be made available, though you will need to keep an eye on such repositories to ensure they are properly patched when vulnerabilities arise and are used only when absolutely necessary. Always prefer packages from Ubuntu's default repositories as well as Snaps, but PPAs offer you another option in case you can't find what you need anywhere else.

After you've maintained a server for a while, or finished setting it up for a particular goal, you'll have installed a plethora of packages to suit its purpose. Exporting a list of installed packages can make it easier to rebuild a server should the need arise, and we'll look at one method of doing that in the next section.

Backing up and restoring Debian packages

As you maintain your server, your list of installed packages will grow. If, for some reason, you needed to rebuild your server, you would need to reproduce exactly what you had installed before, which can be a pain. It's always recommended that you document all changes made to your server via a change control process, but, at the very least, keeping track of which packages are installed is an absolute must. In some cases, a server may only include one or two extra packages in order to meet its goal, but, in other cases, you may need an exact combination of software and libraries in order to get things working like they were. Thankfully, the `dpkg` command allows us to export and import a list of packages to install.

To export a list of installed packages, we can use the following command:

```
dpkg --get-selections > packages.list
```

This command will dump a list of package selections to a standard text file. If you open it, you'll see a list of your installed packages, one per line. A typical line within the exported file will look similar to the following:

```
tmux install
```

With this list, we can import our selections back into the server if we need to reinstall Ubuntu Server, or into a new server that will serve a similar purpose. First, before we manage any packages, we should update our index:

```
sudo apt update
```

Next, we'll need to ensure we have the `dselect` package installed. The `dselect` package provides us with additional features when managing Debian packages. Its finer points are beyond the scope of this chapter, but for our current goal, we can use it to restore packages from our exported list. At your shell prompt, type `which dselect` and you should see output similar to the following:

```
/usr/bin/dselect
```

If you don't see the output, you'll need to install the `dselect` package with `apt`:

```
sudo apt install dselect
```

Once that's complete, you can now import your previously saved package list, and have the missing packages reinstalled on your server. The following commands will complete the process:

```
sudo dselect update
sudo dpkg --set-selections < packages.list
sudo apt-get dselect-upgrade
```

Normally, we simply use `apt` instead of `apt-get` nowadays, but oddly enough, the `dselect-upgrade` command only works with `apt-get`.

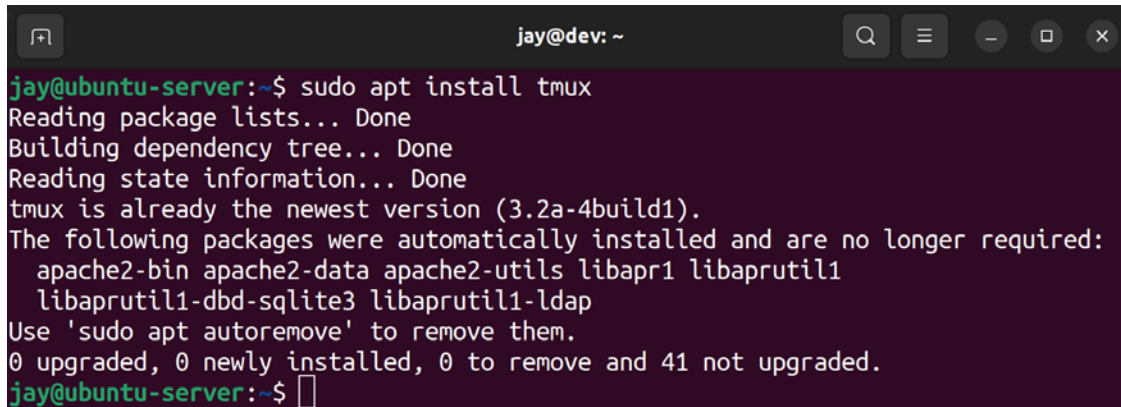
After you have run those commands, the packages that are contained in your packages list but aren't already installed will be installed once you confirm the changes. This method allows you to easily restore the packages previously installed on your server, if for some reason you need to rebuild it, as well as setting up a new server to be configured in a similar way to an existing one.

Now that we have an understanding of how to export and import a list of installed packages, we can also take a look at how to clean up unneeded packages to ensure our server is as free as possible from unnecessary bloat.

Cleaning up orphaned apt packages

As you manage packages on your server, you'll eventually run into a situation where you'll have packages on your system that are installed but not needed by anything. This occurs either when removing a package that has dependencies, or when the dependencies on an installed package change. As you'll remember, when you install a package that requires other packages, those dependencies are also installed. But if you remove the package that required them, the dependencies will not be removed automatically.

To illustrate this situation, if I remove the `apache2` package from one of my servers, I will see the following extra information if I then try to install something else:



```
jay@ubuntu-server:~$ sudo apt install tmux
Reading package lists... Done
Building dependency tree... Done
Reading state information... Done
tmux is already the newest version (3.2a-4build1).
The following packages were automatically installed and are no longer required:
  apache2-bin apache2-data apache2-utils libapr1 libaprutil1
  libaprutil1-dbd-sqlite3 libaprutil1-ldap
Use 'sudo apt autoremove' to remove them.
0 upgraded, 0 newly installed, 0 to remove and 41 not upgraded.
jay@ubuntu-server:~$
```

Figure 3.5: Output with orphaned packages shown

In this example, I removed `apache2` (that was done before the screenshot was taken), then I went on to install `tmux`. The package I was trying to install is arbitrary; the important part is the text you see in the screenshot where it says `The following packages were automatically installed and are no longer required`. Basically, if you have orphaned packages on your system, Ubuntu will remind you periodically as you use the APT suite of tools. In this case, I removed `apache2` so all of the dependencies that were installed to support the `apache2` package were no longer needed.

In the screenshot, I'm shown a list of packages that APT doesn't think I need anymore. It may be right, but this is something we would need to investigate. As instructed in the output, we can use the `apt autoremove` command as root or with `sudo` to remove them. This is a great way of keeping our installed packages clean but should be used with care. If you've just recently removed a package, it's probably safe to do the cleanup manually.

Although we haven't walked through updating packages (we'll do that in *Chapter 21, Securing Your Server*), a situation that may come up later is one in which you have outdated kernels that can be cleaned with the `autoremove` option. These will appear in the same way as the example orphans I was shown in the previous screenshot, but the names will contain `linux-image`. Take care with these; you should never remove outdated kernels from your server until you verify that the newly installed kernel is working correctly and the server doesn't exhibit any unwanted or unexpected behavior. Generally, you would probably want to wait at least a week before running `apt autoremove` when kernel packages are involved.

When it comes to other packages, they are generally safe to remove with the `apt autoremove` command, since the majority of them will be packages that were installed as a dependency of another package that is no longer present on the system. However, double-check that you really do want to remove each of the packages before you do so. You can always reinstall a package if you didn't mean to remove it, and as an added benefit, if you reinstall a package that was marked for auto-removal, it won't show up in the output as an orphan package in the future.

As you've seen so far, there are a lot of options when it comes to managing software in Ubuntu, and as an administrator, you are able to choose the best possible method for your goal. Another special type of update is available, that can improve hardware support in a situation where your hardware is newer than your Ubuntu version, or provide compatibility for hardware that didn't previously have support available.

Taking advantage of hardware enablement updates

One issue in the Linux industry has been hardware support. This is problematic in various Linux distributions because you may find yourself in a situation where you're using a server (or even a desktop or laptop) that was released with the latest processor and chipset, but no newer version of your Linux distribution has been released yet that includes updated drivers that support it. Unlike platforms such as Windows, hardware drivers are typically built right into the Linux kernel. So, if you have an old release (which would contain an older kernel), you might be out of luck for hardware support until the next version of your Linux distribution is released.

Thankfully, Ubuntu has come up with a system to address this problem, and it's one of the many things that set it apart from other distributions. Ubuntu features a set of updates known as the **hardware enablement (HWE)** stack, which is an exclusive feature of **long-term support (LTS)** releases. We discussed the difference between LTS and regular releases back in *Chapter 1, Deploying Ubuntu Server*. HWE updates are optional, but they add additional compatibility with newer hardware that was released after the current LTS version was made available. A new HWE stack typically includes a new kernel and driver packages. However, since drivers are often built right into the Linux kernel, you'll also get added support for things other than new video cards as well, such as newly released network cards.

HWE updates are generally made available from the second point release of an LTS version, and then again with each subsequent point release until the next LTS release becomes available. The new compatibility stack is **backported** from the most recent non-LTS release, which means that you get the same compatibility of the latest non-LTS release while being able to stay on an LTS release. For example, when Ubuntu 20.04.2 was released on February 4, 2021, it contained the kernel and driver packages backported from Ubuntu 20.10. This means that you were able to take advantage of the newer hardware support of Ubuntu 20.10 while being able to remain on 20.04 LTS and benefit from the longer support cycle of the LTS release.

At the time of writing, Ubuntu 22.04 is the latest LTS release of Ubuntu, so hardware enablement hasn't been added yet. Historically, it's most likely the case that hardware enablement will play out in 22.04 LTS the same way as it has in previous releases. If that's the case, then the updated packages will be made available once Ubuntu 22.04.2 is released.

Once the new HWE stack is made available, you can choose to install it or to remain on the original 22.04 kernel with no change. On the desktop version of Ubuntu, people that enjoy computer games can really benefit from these updates, as the new drivers enable better performance when it comes to video cards and support newer gaming hardware. When it comes to the scope of this book, the new hardware enablement benefits us a bit less than desktop users. The reason is that if your server is running fine with no issues, then there really isn't any reason to install the new hardware enablement unless the updated kernel contains new features you'd like to take advantage of.

A frustrating experience that sometimes comes up is having Ubuntu complain that there are no network cards available, even though you do actually have one. I've seen this most often when purchasing a new physical server that contains the latest hardware. It can be very frustrating to power on a brand-new server, only to realize you can't get a network connection at all. This is a classic symptom of needing an updated compatibility stack. Perhaps your network card was released after the release date of the current Ubuntu release. This scenario is a good example of why hardware enablement updates exist, and why you will probably end up using them sooner or later.

If you don't opt in to the HWE updates, your server will always have the same hardware enablement (kernel, drivers, and so on) as it did when your installed LTS release was first published. In that case, your kernel and related packages will only be updated when you install new security updates. At a later date, you can opt in to HWE updates manually if you wish. Generally, you only do this if you've added new hardware to a physical server that requires a new kernel. If your server is working perfectly fine and you haven't added new hardware, there's probably no reason to install a new HWE stack.

If you do decide to utilize these updates, there are two ways to do so. You can opt in to the newer HWE stack while installing Ubuntu Server, or you can manually install the required packages. At the time of writing, Ubuntu 22.04 is new to the scene, so it remains to be seen if the updated packages will be made available with 22.04.2, but it's very likely given Ubuntu's history. The following screenshot shows the first screen of the installer for Ubuntu 20.04.4. Assuming Ubuntu 22.04 follows the same plan, you'll most likely see an option such as the one in the following screenshot once 22.04.2 is released:

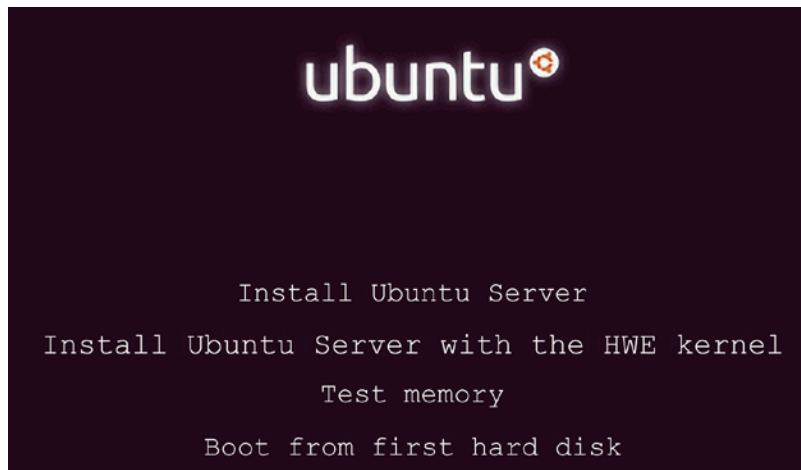


Figure 3.6: Main menu of the installer for Ubuntu 20.04.4

Notice the **Install Ubuntu Server with the HWE kernel** option. If you choose this option, then your installation of Ubuntu Server will contain the new HWE packages immediately.

If you've already installed Ubuntu Server and you'd like to install the HWE updates afterward, you can do so from the terminal. In Ubuntu 20.04, for example, you were able to switch to the HWE kernel with the following command:

```
sudo apt install --install-recommends linux-generic-hwe-20.04
```

When Ubuntu 22.04 releases its newer HWE stack, a similar command will likely be used to install it. If you need a newer HWE kernel, refer to the instructions in Ubuntu's documentation pages when that time comes.

Summary

In this chapter, we have taken a crash course in the world of package management. As you can see, Ubuntu Server offers an amazing number of software packages and various tools that we can use to manage them. We began the chapter with a discussion of how package management with Ubuntu works, then we worked through installing packages, searching for packages, and managing repositories. Snap packages were also covered, which is a newer technology that aims to enhance software distribution on Ubuntu.

In *Chapter 4, Navigating and Essential Commands*, we'll take a look at foundational commands for navigating the Linux Shell, understanding the filesystem layout, and more.

Relevant video

- Linux Crash Course - APT (LearnLinuxTV): <https://linux.video/lcc-apt>

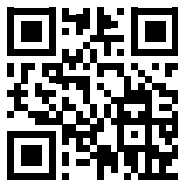
Further reading

- LTSEnablementStack (Ubuntu wiki): <https://learnlinux.link/lts-es>
- Ubuntu Server Guide: <https://ubuntu.com/server/docs>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



4

Navigating and Essential Commands

At this point in our journey, we've already covered a lot of ground—we've learned how to deploy an Ubuntu server, how to manage users, and most recently, how to manage software packages. Before we go too far, we should take a moment to learn some important concepts and commands that will allow us to build more of the foundational knowledge that will serve us well for the remainder of the book and beyond. These foundational concepts include core Linux commands for navigating the shell, the Linux filesystem layout, viewing the contents of files, and even checking on log files. Specifically, this discussion will include:

- Learning essential Linux commands
- Understanding the Linux filesystem layout
- Viewing the contents of files
- Viewing application log files

Let's take some time to learn some essential Linux commands that will help strengthen our command-line skills.

Learning essential Linux commands

Building a solid competency on the command line is essential and effectively gives any system administrator or engineer superpowers. Our new abilities won't allow us to leap tall buildings in a single bound but will definitely enable us to execute terminal commands as if we're command-line ninjas. While we won't completely master the art of using the command line in this section (that can only come with years and experience), we will definitely become more confident.

First, let's talk about moving from one place to another within the Linux filesystem. Specifically, by "Linux filesystem," I'm referring to the default structure of the various folders (also referred to as "directories") contained within your Ubuntu installation. The Linux filesystem contains many important directories, each with its own designated purpose, which we'll talk about in more detail later in this chapter.

Before we can explore that further, we'll need to learn how to navigate from one directory to another. The first command we'll cover in this section relative to navigating the filesystem will clarify the directory you're currently working from. For that, we have the `pwd` command. This stands for **print working directory** and shows you where you currently are in the filesystem. If you run it, you may see output such as this:

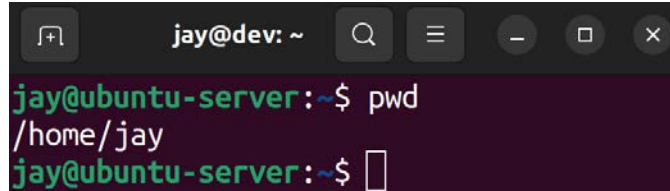
A terminal window titled 'jay@dev: ~' with search, menu, and window control icons. The prompt is 'jay@ubuntu-server:~\$'. The user enters 'pwd' and the output is '/home/jay'. The prompt returns to 'jay@ubuntu-server:~\$' with a cursor.

Figure 4.1: Viewing the current working directory

In this example, when I ran `pwd`, the output informed me that my current working directory is `/home/jay`. This is known as your home directory and, by default, every user has one (as we discussed in *Chapter 2, Managing Users and Permissions*). This is where all the files for your user account will reside by default. Sure, you can create files anywhere you'd like, even outside your home directory if you have permission to do so or you use `sudo`. But just because you *can* doesn't mean you *should*. As you'll learn in this chapter, the Linux filesystem has a designated place for just about everything. But your home directory, located at `/home/<username>`, is yours. You own it, you control it—it's your home on the server. In the early 2000s, Linux installations with a graphical user interface even depicted your home directory with an icon of a house.

Typically, files that you create in your home directory will have permission strings similar to this:

```
-rw-rw-r-- 1 jay jay    0 Jul  5 14:10 testfile.txt
```

We've discussed permissions and gone over how to read a permission string in *Chapter 2, Managing Users and Permissions*, but you can see that by default, files you create in your home directory are owned by your user, your group, and are readable by all three categories (user, group, and other).

To change our current directory and navigate to another, we can use the `cd` command along with a path we'd like to move to:

```
cd /etc
```

Now, I haven't gone over the file and directory layout yet, so I just randomly picked the `/etc` directory. The forward slash at the beginning designates the beginning of the filesystem. More on that later. Now, we're in the `/etc` directory, and our command prompt has even changed as well:

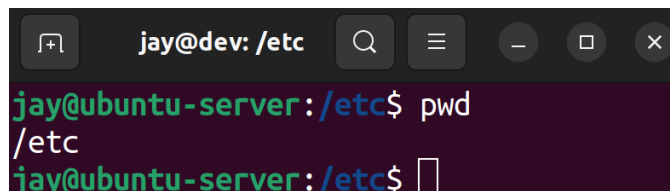
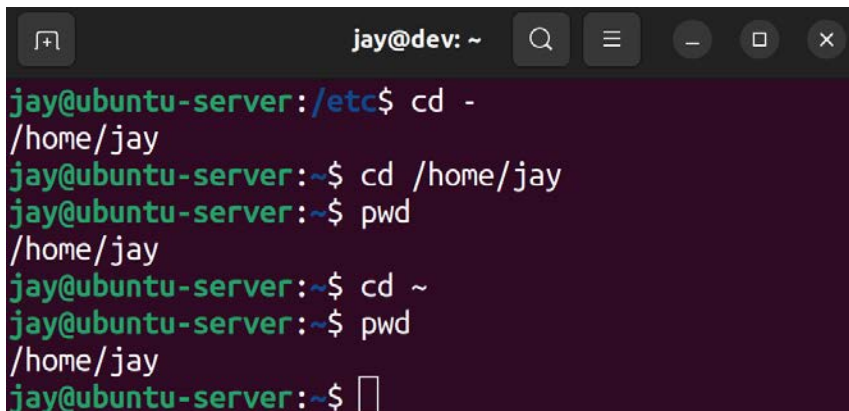
A terminal window titled 'jay@dev: /etc' with search, menu, and window control icons. The prompt is 'jay@ubuntu-server:/etc\$'. The user enters 'pwd' and the output is '/etc'. The prompt returns to 'jay@ubuntu-server:/etc\$' with a cursor.

Figure 4.2: Command prompt and `pwd` command after changing a directory

As you could probably guess, the `cd` command stands for *change directory*, and it's how you move your working directory from one to another while navigating around. You can use the following command, for example, to return back to the home directory:

```
cd /home/<user>
```

In fact, there are several ways to return home, a few of which are demonstrated in the following screenshot:



```
jay@dev: ~
jay@ubuntu-server:~/etc$ cd -
/home/jay
jay@ubuntu-server:~$ cd /home/jay
jay@ubuntu-server:~$ pwd
/home/jay
jay@ubuntu-server:~$ cd ~
jay@ubuntu-server:~$ pwd
/home/jay
jay@ubuntu-server:~$
```

Figure 4.3: Other ways of navigating to the home directory

The first command, `cd -`, doesn't actually have anything to do with your home directory specifically. It's a neat trick to return you to whatever directory you were in most recently. For me, the `cd -` command took me to the previous directory I was in, which just so happened to be `/home/jay`. The second command, `cd /home/jay`, took me directly to my home directory since I called out the entire path. The last command, `cd ~`, also took me to my home directory. This is because `~` is shorthand for the full path to your home directory, so you don't really ever have to type out the entire path to `/home/<user>`. You can just refer to that path simply as `~`.

Another essential command is `ls`. The `ls` command lists the contents of the current working directory. We probably don't have any content in our home directory yet. But if we navigate to `/etc` by running `cd /etc`, as we did earlier, and then execute `ls`, we'll see that the `/etc` directory has a number of files in it. Go ahead and try it yourself and see:

```
cd /etc
ls
```

We didn't actually have to change our working directory to `/etc` just to list the contents. We could've just executed the following command:

```
ls /etc
```

Even better, we can run:

```
ls -l /etc
```

This gives us the contents in a long list, which I think is much easier to understand. It will show each directory or file entry on its own line, along with the permission string. But, you probably already remember `ls` as well as `ls -l` from back in *Chapter 2, Managing Users and Permissions*, so I won't go into too much more detail here. The `-l` portion of the `ls` command in that example is known as an **argument**. I'm not referring to an argument such as the ever-ensuing debate in the Linux community over which command-line text editor is the best between Vim and Emacs (it's clearly Vim). Instead, I'm referring to the concept of an argument in shell commands that allow you to override the defaults, or feed options to the command in some way, such as in this example, where we format the output of `ls` to be in a long list.

The `rm` command is another one that we touched on in *Chapter 2, Managing Users and Permissions*, when we were discussing manually removing the home directory of a user that was removed from the system. So, at this point, you're probably well aware of that command and what it does (it removes files and directories). It's a potentially dangerous command, as you could use it to accidentally remove something that you shouldn't have. In that chapter, we used the following command to remove the home directory of user `dscully`:

```
rm -r /home/dscully
```

As you can see, we're using the `-r` argument to alter the behavior of the `rm` command, which, by default, doesn't remove directories but only files. The `-r` argument instructs `rm` to remove everything recursively, even if it's a directory. The `-r` argument will also remove subdirectories of the path as well, so you'll definitely want to be careful with this command. As I've mentioned earlier in the book, if you use `sudo` with `rm`, you can hypothetically delete your entire Ubuntu installation!

Another option offered by `rm` is the `-f` argument, which is short for *force*, and it tells `rm` not to prompt before removing things. This argument won't be needed as often, and use cases for it are outside the scope of this chapter. But keep in mind that it exists, should you need it.

Another foundational command that's good to know is `touch`, which actually serves two purposes. First, assuming you have permission to do so in your current working directory, the `touch` command will create an empty file if it doesn't already exist. Second, the `touch` command will update the modification time of a file or directory if it does already exist:



```
jay@dev: ~  
jay@ubuntu-server:~$ touch testfile.txt  
jay@ubuntu-server:~$ ls -l  
total 0  
-rw-rw-r-- 1 jay jay 0 May  9 04:18 testfile.txt  
jay@ubuntu-server:~$ touch testfile.txt  
jay@ubuntu-server:~$ ls -l  
total 0  
-rw-rw-r-- 1 jay jay 0 May  9 04:18 testfile.txt  
jay@ubuntu-server:~$
```

Figure 4.4: Experimenting with the `touch` command

To illustrate this, in the related screenshot, I ran several commands. First, I ran the following command to create an empty file:

```
touch testfile.txt
```

That file didn't exist before, so when I ran `ls -l` afterward, it showed the newly created file with a size of 0 bytes. Next, I ran the `touch testfile.txt` command again a minute later, and you can see in the screenshot that the modification time went from 15:12 to 15:13.

When it comes to viewing the contents of a file, we'll get to that later on in this chapter. And there are definitely more commands that we'll need to learn to build the basis of our foundation. But for now, let's take a break from the foundational concepts to understand the Linux filesystem layout better, which is pretty much mandatory for some of the commands we'll learn later.

Understanding the Linux filesystem layout

As I mentioned earlier, every directory in a Linux installation has a designated purpose. It isn't a hard rule that you have to follow, more of a strong recommendation for where certain things are supposed to go.

You can certainly go against the recommendations; ultimately, you have full control over your installation. But, if you make it a habit to place files in strange locations, you may annoy your colleagues. In this section, we'll go over the most common directories and talk about their purpose.

The term *filesystem* itself can be somewhat confusing in the Linux world because it can refer to two different things—the default directory structure, as well as the actual filesystem we choose when formatting a volume such as a hard drive or flash drive (ext4, XFS, and so on). Specific to this section, we're going to take a quick look at the Linux filesystem in the context of the default directory structure.

In Linux (Ubuntu uses the Linux kernel and related utilities, so therefore it is a *distribution* of Linux), the filesystem begins with a single forward slash, `/`. This is considered the beginning of the filesystem, and directories and sub-directories branch out from there. For example, consider the `/home` directory. This directory exists at the root level of the filesystem, which you can see from the fact that it begins with a forward slash. The home directory on my system is `/home/jay`, which means that it's the `jay` directory, which is inside the home directory, and that directory is at the beginning of the filesystem.

This is confusing at first but becomes very logical once you become accustomed to it. If you're familiar with Microsoft Windows, then you can technically think of `/` as the `C:` drive. It's actually a bit more complex than that, but if we forego some of the quirks, that comparison works. To really bring this home, use the `ls` command against several directories on your server. If you execute `ls /`, you will see all of the directories at the root of the filesystem.

You'll see the home directory among the results, among many others. For those of you who prefer a more visual representation, the following screenshot shows an example filesystem:

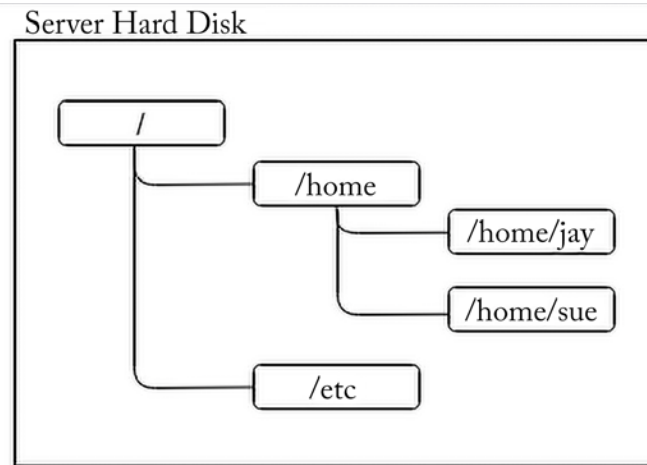


Figure 4.5: Diagram of a portion of a typical Linux filesystem

As you can see, the Linux filesystem resembles a tree, with a main branch that extends outward and directories branch out from one another. This default directory structure is part of the **Filesystem Hierarchy Standard (FHS)**, which is a set of guidelines that defines how the directory structure is laid out. This specification defines the names of the directories, where they are located, and what they are for. Distributions will sometimes go against some of the definitions here, but for the most part, follow it fairly closely. This is why you may see a very similar (if not the same) directory structure on distributions of Linux other than Ubuntu.

So, why is this important? As I mentioned, each directory generally has a purpose. There are some debates about the default layout from time to time, and some changes are made every now and then. But in regard to Linux, the filesystem layout tends to change less frequently than other things do.

A full walkthrough of the FHS would be astronomical, but I have included a link to this specification at the end of the chapter should you decide to read more about it. There are some directories you definitely should know, however. Here are some of the more important ones.

Directory	Purpose
/	The beginning of the filesystem; all directories are underneath this
/etc	System-wide application configuration
/home	User home directories
/root	The home directory for root (root doesn't have a directory under /home)
/media	For removable media, such as flash drives
/mnt	For volumes that are intended to stay mounted for a while
/opt	Additional software packages (some programs are installed here, not as common)

/bin	Essential user binaries (<code>ls</code> , <code>cp</code> , and so on)
/proc	Virtual filesystem for OS-level components
/usr/bin	A majority of user commands
/usr/lib	Libraries
/var/log	Log files

The `/etc` directory deserves some additional discussion, as you'll no doubt be working with it a lot. This directory, as mentioned in the preceding table, holds configuration files for applications that are intended to be respected system-wide. For example, if you're running the OpenSSH daemon on your server, you're listening for connections via port 22 by default. (Don't worry, we'll discuss these individual concepts in further chapters.) The configuration file for the OpenSSH server is located in the `/etc/ssh` directory. Since OpenSSH is a service that runs on the system as a background process, its configuration is stored in the `sshd_config` file inside that directory. If you remove the package responsible for providing OpenSSH, the configuration is retained in that file (removing packages doesn't also remove config files by default), so if you go to reinstall OpenSSH later, you'll have the same configuration the next time. If we want to wipe out the configuration while uninstalling a package, we can use the `--purge` option with the `apt remove` command to do so, as we saw in *Chapter 3, Managing Software Packages*.

Other directories of importance will be discussed as we progress through the topics in this book. Don't worry if some of this doesn't make sense right now; it will come with time. The main point here is that there are many directories, each having its own purpose. If you want to know the purpose of a particular directory, consult the FHS. If you're curious where you should place something on a server, also consult the FHS. But again, don't worry too much about doing research outside of this book, as we'll cover the necessary topics as we go along.

We now have a better understanding of the default filesystem layout and the purpose of some common directories. In the next section, we'll explore how we can view the contents of files stored in those directories.

Viewing the contents of files

The Linux filesystem contains many directories and files. In the case of files, we need to learn how to read and manipulate them to round out our knowledge. We'll cover more topics surrounding file management in the next chapter. For now, we can benefit by taking a look at how to view the content of existing files.

We can print the contents of a file to the screen with the `cat` command, along with a filename as an argument. The following command, for example, can be used to view the contents of `testfile.txt` in our current working directory, which we created earlier in the chapter when we discussed the `touch` command. Sure, this exercise is rather pointless since that file is empty, but it offers a good first example:

```
cat testfile.txt
```

There's no output, since again, the file is empty. So, let's instead take a look at a more practical example. Here's a file we can use `cat` against that actually does have content inside:

```
cat /etc/os-release
```

The output of this command is as follows:

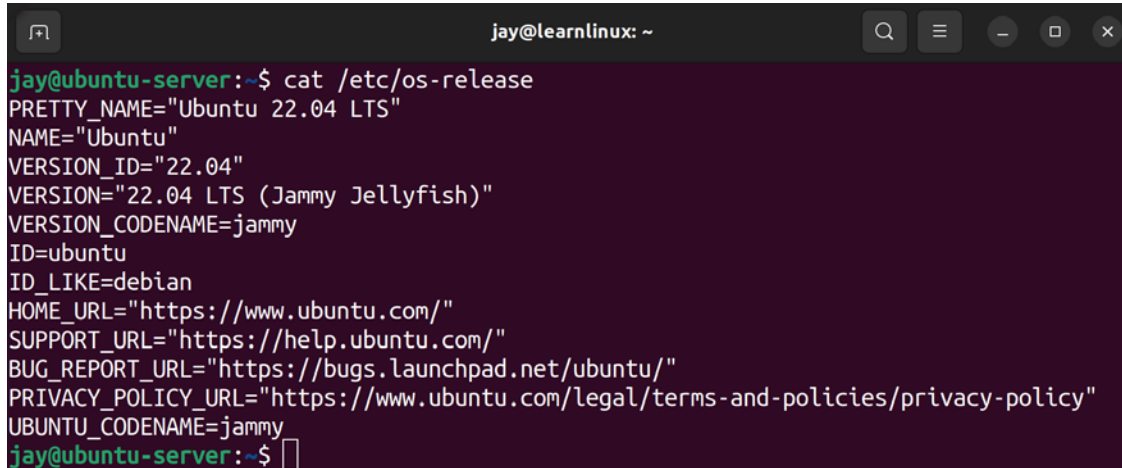
A terminal window titled 'jay@learnlinux: ~' with search, menu, and window control icons. The prompt is 'jay@ubuntu-server:~\$'. The command 'cat /etc/os-release' has been executed, resulting in the following output: 'PRETTY_NAME="Ubuntu 22.04 LTS"', 'NAME="Ubuntu"', 'VERSION_ID="22.04"', 'VERSION="22.04 LTS (Jammy Jellyfish)"', 'VERSION_CODENAME=jammy', 'ID=ubuntu', 'ID_LIKE=debian', 'HOME_URL="https://www.ubuntu.com/"', 'SUPPORT_URL="https://help.ubuntu.com/"', 'BUG_REPORT_URL="https://bugs.launchpad.net/ubuntu/"', 'PRIVACY_POLICY_URL="https://www.ubuntu.com/legal/terms-and-policies/privacy-policy"', and 'UBUNTU_CODENAME=jammy'. The prompt returns to 'jay@ubuntu-server:~\$'.

Figure 4.6: Viewing the contents of `/etc/os-release`

The `/etc/os-release` file is one that exists on many distributions. It's a special file that gives you some information about the distribution of Linux that's currently installed. If you were to connect to a Linux server and wanted to know which distribution it was running, viewing the contents of this file is one way to find out. You can also view some of the same info in abbreviated form via the following command:

```
lsb_release -a
```

That command also works on various distributions, but I prefer the `/etc/os-release` file because it contains more information. Regardless, the entire point of this exercise is to demonstrate that the `cat` command allows you to view the contents of a file. More or less, there are other commands that also let you do the same thing. And I mean that literally—you can also try viewing the contents of `/etc/os-release` with `more` or `less`:

```
more /etc/os-release  
less /etc/os-release
```

The `more` command allows you to more easily view larger files, so the benefit won't be as immediately apparent with a file as short as `/etc/os-release`. If you use `more` to view a longer file, it will stop the output when it fills the screen, and allow you to press *Enter* to advance to the next line.

The `less` command allows you to do the same thing, but also allows you to not only use your arrow keys in addition to *Enter* to view more output; it also allows you to advance forward or backward as well. So essentially, the `less` command gives you more features than the `more` command does.

In addition, we also have the `grep` command at our disposal. It's not typically used to simply view the contents of a file, but it's definitely a great command to know that can help you view specifically what you want to view, rather than the entire file.

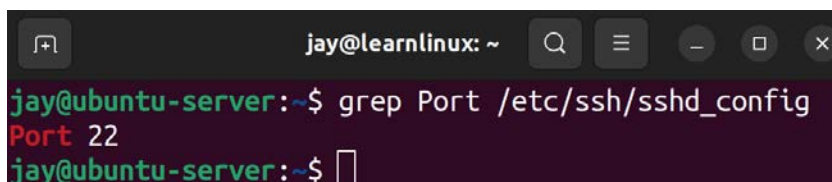
If you chose to install the OpenSSH server when you first installed Ubuntu Server, you should have the config file for it in your installation, and you can view the contents of that file with `cat` as we normally would:

```
cat /etc/ssh/sshd_config
```

Of course, that's going to dump the contents of that file onto our screen, and the file is many more lines than what the typical monitor is able to display all at once. We might be interested in a particular line or word, so we'll need to be able to narrow down the file to what we actually care about. We'll talk about the OpenSSH server in more detail in *Chapter 10, Connecting to Networks*, so don't worry about what this configuration file means yet. Let's just say, hypothetically, we're only interested in the port that OpenSSH is listening on. We can use the `grep` command to try and print only the lines of the `/etc/ssh/sshd_config` file that pertain to that specific configuration:

```
grep Port /etc/ssh/sshd_config
```

This command will produce the following output:

A terminal window screenshot with a dark background. The prompt is 'jay@ubuntu-server:~\$'. The command 'grep Port /etc/ssh/sshd_config' has been entered. The output 'Port 22' is displayed in red text. The prompt 'jay@ubuntu-server:~\$' is shown again with a cursor at the end of the line.

```
jay@ubuntu-server:~$ grep Port /etc/ssh/sshd_config
Port 22
jay@ubuntu-server:~$
```

Figure 4.7: Viewing the contents of the `/etc/ssh/sshd_config` file with `grep` to find lines containing “Port”

Essentially, what we're doing is instructing `grep` to print only the lines of the `/etc/ssh/sshd_config` file that contain the string `Port`. In the screenshot, a single line contained a match for that string, so it displayed. That output is certainly better than scrolling through the 123 lines of text in the file, when we're only interested in lines pertaining to the port.

By default, `grep` is case-sensitive. This means that if we were to use `grep` to find lines matching “port” (with a lowercase P), we would get no output at all. We can simply add the `-i` argument to make the search case-insensitive.

It's very common to see the `grep` command paired with another command, such as `cat`:

```
cat /etc/ssh/sshd_config | grep Port
```

That's a perfectly valid command and will do the same thing. However, it's fairly redundant. We'll get into the concept of redirecting output in the next chapter, but essentially this command takes the output of the `cat` command and redirects it as input to the `grep` command. Using `cat` to first print out the file and then have `grep` grab the contents of that file and search it for a string is a two-step process where only one step is required. But again, it's still a valid command.

To this day, I'll personally use the `cat` command with `grep` to do the same thing out of habit, as this was how all new Linux users were taught back when I started. You'll even see me do this in my YouTube videos—old habits are hard to break!

Log files are a great source of information about what's going on in the background of your server, and are especially useful for troubleshooting. We'll now transition into a discussion on viewing these files and some relevant commands pertaining to logging.

Viewing application log files

In the last section of this chapter, let's explore log files a bit, as they bring several concepts that we've discussed full circle. We went through an overview of the default directory layout, practiced viewing files, and we learned how to search files for strings. We'll discuss log files in greater detail later on in the book, but we can use all of these concepts to take an initial look at viewing log files now.

If you recall, during the discussion of the Linux filesystem layout earlier in this chapter, there was a table showing some of the most common directories that exist. Among the items in that table, I called out the `/var/log` directory. While logging is transitioning to a different style (more on that in *Chapter 22, Troubleshooting Ubuntu Servers*), we'll have a series of logs in the `/var/log` directory. Go ahead and use the `ls` command yourself, and you'll see there are quite a few files there. While I won't go through all of them in this chapter, let's take a look at `/var/log/syslog`.

The content of this file is going to have quite a few lines. This is the system log, which is used to view informational messages about what's going on in the background as Ubuntu runs on your server, and will show warnings and errors as well. If you run into a problem where something isn't working quite right, you may see output in the system log that will provide you with some sort of error you can look up in a search engine so you can try and find a resolution. For example, you can use `grep` to view any lines that contain the search term *Network* if you're having issues with your connection:

```
grep Network /var/log/syslog
```

That's just a hypothetical example, but it may show lines that are relevant. Adjust your search term to whatever you're interested in finding.

This is also a good time to introduce you to the `head` and `tail` commands. These commands will show the first ten or last ten lines of a file, respectively. This is useful for the `/var/log/syslog` file because again, that file is quite large, and you may only be interested in certain lines. You can also adjust the number of lines the `head` and `tail` commands show with the `-n` option with a desired number of lines. For example, to view the last 100 lines of a file:

```
tail -n 100 /var/log/syslog
```

Perhaps even more useful is the `-f` option:

```
tail -f /var/log/syslog
```

This allows you to follow (watch) a file in (almost) real time. With the `-f` option, the terminal will continue to display new lines in this file as they're appended, so you can monitor the log file as someone attempts to reproduce a problem, for example. You can press `Ctrl + c` on your keyboard to break out of the follow mode and return back to the command prompt.

Of course, there are many more foundational commands and concepts that we can go over, but I think that this is enough for now. In the next chapter, we'll expand on this further. But for now, I recommend you practice all of the concepts in this chapter until you're familiar with them before we go on.

Summary

There are more Linux commands than you'll ever be able to memorize. Most of us just memorize our favorite commands and variations of commands, and you'll develop your own menu of these commands as you learn and expand your knowledge. In this chapter, we covered many of the foundational commands that are, for the most part, essential. Commands such as `grep`, `cat`, `cd`, `ls`, and others were explored this time around. The next chapter is essentially a continuation of this one, but I wanted to split the foundational concepts into two chapters rather than one enormous one.

In the next chapter, we'll expand our foundational knowledge with a deeper look at file management, including editing files, input/output streams, and symbolic links, and we'll even reveal the secret to life itself. Well, maybe not the latter, but the next chapter will still be great. See you there!

Relevant video

- Linux Crash Course – Navigating the Linux Filesystem (LearnLinuxTV): <https://linux.video/lcc-navigating>

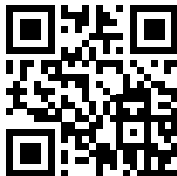
Further reading

- Filesystem Hierarchy Standard: <https://learnlinux.link/fhs-doc>
- UsingTheTerminal: <https://learnlinux.link/using-term>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



5

Managing Files and Directories

In *Chapter 4, Navigating and Essential Commands*, we started looking deeper into Linux commands. We went over the most essential commands and covered the filesystem layout, as well as various methods to view the contents of files. In this chapter (as well as the next), we'll continue to expand on the command line and become more efficient while using the terminal. This time around, we'll expand a bit further on file management, take a look at input/output streams, and we'll also develop an understanding of symbolic links. Along the way, we will cover:

- Copying, moving, and renaming files and directories
- Editing files with the Nano and Vim text editors
- Input and output streams
- Using symbolic and hard links

Let's begin the chapter by taking a look at some methods we can use to alter the files and directories in the filesystem, such as copying and moving them.

Copying, moving, and renaming files and directories

At this point, you should know how to move around within the filesystem (commands such as `cd`), inspect the contents of directories (`ls`), and even how to create empty files (the `touch` command). We even know how to remove files as well, such as executing the `rm` command against a file or directory. But until now, we haven't looked at moving files around within your Ubuntu filesystem.

First, to make a copy of a file or directory, we use the `cp` command. Copying a file is fairly easy, and such a command would look similar to the following:

```
cp file1 file2
```

In that example, `file2` is created as an exact copy of `file1`. Copying a file is useful in many situations, some of the most common of which are:

- Copying a file to a backup medium, such as an external drive or network share
- Creating a copy of a file before making a change, such as before editing a very important text file

- Duplicating a log file for a point-in-time analysis

Let's look at the last bullet point as another good example. We can capture a copy of the system log and store it in our current working directory by executing the following command:

```
sudo cp /var/log/syslog /home/<username>/syslog
```

The `cp` command is fairly simple: we give it a path to a file to copy, and then we type the path to the destination and desired filename. We also use `sudo` due to the fact that the `syslog` may not be readable by normal users by default, depending on the configuration. In the case of this command, a copy of the original `syslog` file will be saved in your current working directory.

For this particular example, log files are constantly being written to. It can sometimes be hard to troubleshoot an issue that occurred during a certain time if the file is continually expanding. That's not the only reason though. We certainly wouldn't want to make an accidental change to the log file and risk contaminating it or losing important information.

The previous command can actually be simplified a bit:

```
sudo cp /var/log/syslog .
```

In that example, we removed the target path and name and replaced it with a period. The reason this works is that the period represents our current working directory. This isn't specific to the `cp` command either. There actually is a directory named with a period in every directory, which is essentially a pointer to the current directory. Therefore, if you're already inside the directory you want the file to be copied to, you don't have to type the path. You don't have to type the name either if you want the name of the file to be the same as the source.

Copying files (as well as moving them, which we'll look at next) is potentially destructive. If the target path and name already exist, then the target will be overwritten. By default, you won't see a confirmation before the target file is overwritten. As with all commands, take care that you really want to do whatever it is you're telling the command-line interpreter to do.

When it comes to copying directories, the `cp` command by itself won't do the trick:

```
sudo cp /var/log/apt .
```

The `/var/log/apt` directory contains log files that keep track of transactions performed with the `apt` command. It can be useful to keep an eye on what your other admins are installing. When it comes to this example though, the previous command will fail with the following error:

```
cp: -r not specified; omitting directory '/var/log/apt'
```

The error literally tells you what to do about it, and essentially the error is telling you that directories are omitted by default. In order to copy directories, you'll need to include the `-r` option. This stands for *recursive*, which is an option you'll see as a possibility for a handful of Linux commands. It tells the command-line interpreter to not only grab the object at the path you specified but to do so recursively and include child objects as well. Therefore, the following command will work:

```
sudo cp -r /var/log/apt .
```

With that command, a local copy of the `/var/log/apt` directory will be stored in your current working directory, along with the contents.

When it comes to moving a file or directory from one place to another, we use the `mv` command. The syntax is almost exactly the same as the `cp` command. The difference is that instead of copying a file or directory, we're moving it. In that regard, it's probably self-explanatory how it works. Consider the following example commands:

```
mv file1 /path/to/new/directory/file1
mv file1 file2
```

With the first example, we're assuming that `file1` is in our current working directory. We're grabbing that file and moving it to `/path/to/new/directory` and giving it the same name of `file1` in that new directory. Just like with the `cp` command, we could've omitted the filename at the target, since it's staying the same. If the file already exists in the target directory with that name, it will be overwritten. So, the `mv` command is also potentially destructive, but more so when compared to `cp` since you are moving, instead of copying.

The second `mv` command is a bit more interesting, because in that example we're renaming a file. In Linux, there's no dedicated rename command, so the `mv` command is used for that purpose. In fact, `mv` is somewhat of a Swiss Army knife due to the fact it serves multiple purposes. With it, you can move a file or directory, rename it, and also move a file to overwrite another file in a different location. It all depends on the source and destination paths. If the target exists, it will be overwritten. If not, the file will be renamed and/or moved to that path.

When managing files, you'll certainly come to a situation where you need to manage multiple files or directories. It's worth noting that the `cp` command, as well as the `mv` command, can be used with multiple objects at once; for example, if you have three directories, such as `dir1`, `dir2`, and `dir3`, and you need to move them into a new sub-directory. You could execute three `mv` commands to move each one separately, but you can also move all three with a single `mv` command:

```
mv dir1 dir2 dir3 /path/to/new/location
```

The same holds true with `cp` and with files; both `cp` and `mv` are set up to allow you to move or copy multiple directories or files with a single command.

Now that we know how to move files around, we should also take a look at how to edit them. There are many text editors available to us on the Ubuntu platform, with Nano and Vim being among the most common. In the next section, we'll go over the basics of both.

Editing files with the Nano and Vim text editors

Now that we know how to copy and move files, it would be useful to know how to edit them. There are multiple forms of text editors for Ubuntu, some available in the command line, and others in graphical environments such as the text editor in the desktop version.

Some may feel that command-line text editors are more complicated than **Graphical User Interface (GUI)** editors (and to be honest, they can be), but the main benefit is that you can use the same editor regardless of whether or not you have a GUI. In a way, this means the non-graphical editors are a bit more portable, and you can rely on them more. Almost all installations of Ubuntu will include the nano text editor, and you can rely on that more often than something like a particular GUI editor being available. In addition, the vim editor is another popular consideration. It's a bit more advanced than nano, but in my opinion, much more powerful. In the following sections, we'll look at both nano and vim.

Editing with Nano

The nano editor, while more basic in terms of features, still has a fairly strong following. Actually, if you haven't already noticed, a person's choice of editor can cause somewhat of a debate in the Linux community. To launch the nano editor, it's as simple as running the nano command. If you don't provide nano with a filename, it simply starts with an empty window, as shown in the following screenshot:

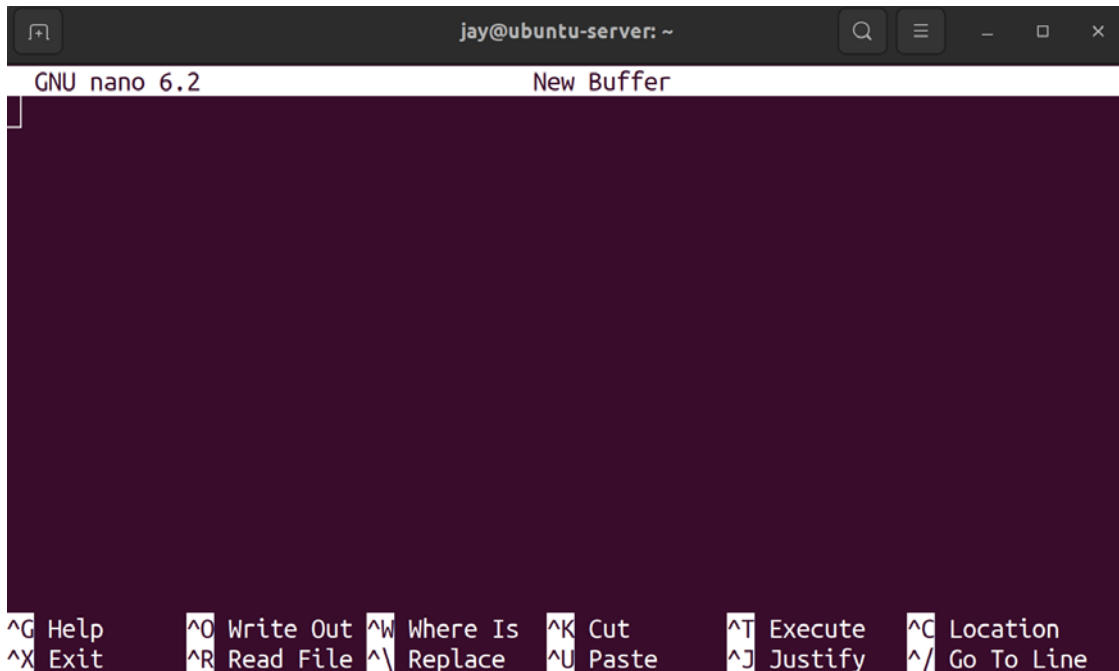


Figure 5.1: The nano text editor, with no file selected

You can start typing right away, and when you want to save the file, you can simply press `Ctrl + o` to do so. On the bottom of the nano screen, you'll see an overview of shortcuts, including the method for saving the file I just mentioned (which it refers to as **Write Out**). As you can see then, `Ctrl + x` is how you exit the editor and go back to the command line.

One trick that I love (which also works well with vim) is pressing `t` and then `z` while holding `Ctrl` when you have nano open, which will make it disappear. Normally, you don't have to press `Ctrl` and `t` in order to background a process, but nano is the exception in this case.

Effectively, it's the same as minimizing it. You can bring the nano window back by executing `fg`, which is short for *foreground*. The ability to background and foreground an application in the terminal is part of process management, which is actually something we'll talk about in *Chapter 7, Controlling and Managing Processes*, but we're giving you this tip ahead of time, to help you on your way! When it comes to editing files, sometimes it makes sense to send the editor to the background and bring it back later.

As you can probably guess, you can also use the nano command, with a target path and filename, to have the editor come up with a file already open. For example:

```
sudo nano /var/log/syslog
```

This command will result in the file opening in nano:

```

GNU nano 6.2 /var/log/syslog
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Linux version 5.15.0-25-generic (bu
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Command line: BOOT_IMAGE=/vmlinuz-5
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] KERNEL supported cpus:
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Intel GenuineIntel
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] AMD AuthenticAMD
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Hygon HygonGenuine
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Centaur CentaurHauls
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] zhaoxin Shanghai
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] x86/fpu: x87 FPU will use FXSAVE
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] signal: max sigframe size: 1440
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] BIOS-provided physical RAM map:
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] BIOS-e820: [mem 0x0000000000000000->
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] BIOS-e820: [mem 0x00000000000009fc00->
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] BIOS-e820: [mem 0x000000000000f0000->
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] BIOS-e820: [mem 0x0000000000100000->
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] BIOS-e820: [mem 0x0000000007ffda000->
[ File '/var/log/syslog' is unwritable ]
^G Help      ^O Write Out ^W Where Is  ^K Cut       ^T Execute  ^C Location
^X Exit      ^R Read File ^_ Replace   ^U Paste     ^J Justify  ^/_ Go To Line

```

Figure 5.2: The nano text editor, with the syslog file open

To be fair, editing a log file directly is probably not a good idea in practice, but the example works. The syslog is a file you may want to open and inspect, but it would be a good idea to check the size of the syslog file with the `ls -lh /var/log/syslog` command to make sure it's not excessively large in size, which can slow down the server.

All of that aside, the takeaway here is that you can open a file with nano directly by simply providing it with a path to something. Inside the editor window, you can move around with the arrow keys, just like you would in a graphical editor. You can also search for a specific string of text, by pressing `Ctrl + w` for **Where Is**, as is mentioned in the list of actions near the bottom.

In addition, note the text in the screenshot that is highlighted in red. It's simply telling us that we don't have access to edit the file. In that example, I didn't use sudo because I didn't intend on making any changes to the file.

Sometimes, I'll open files in an editor that I don't have write permission to make changes to, which serves as a fail-safe in case I accidentally do make a change (I won't be able to save the file without `sudo`). To make a real change, I can close the file and reopen it with `sudo`.

Anyway, the nano editor is fairly simplistic. Is there more to it than that? Sure, but the most important thing to know is how to open files, edit them, and save them, which we've covered in this section. Of course, you can practice with the actions that are presented to you at the bottom of the window to go beyond normal usage.

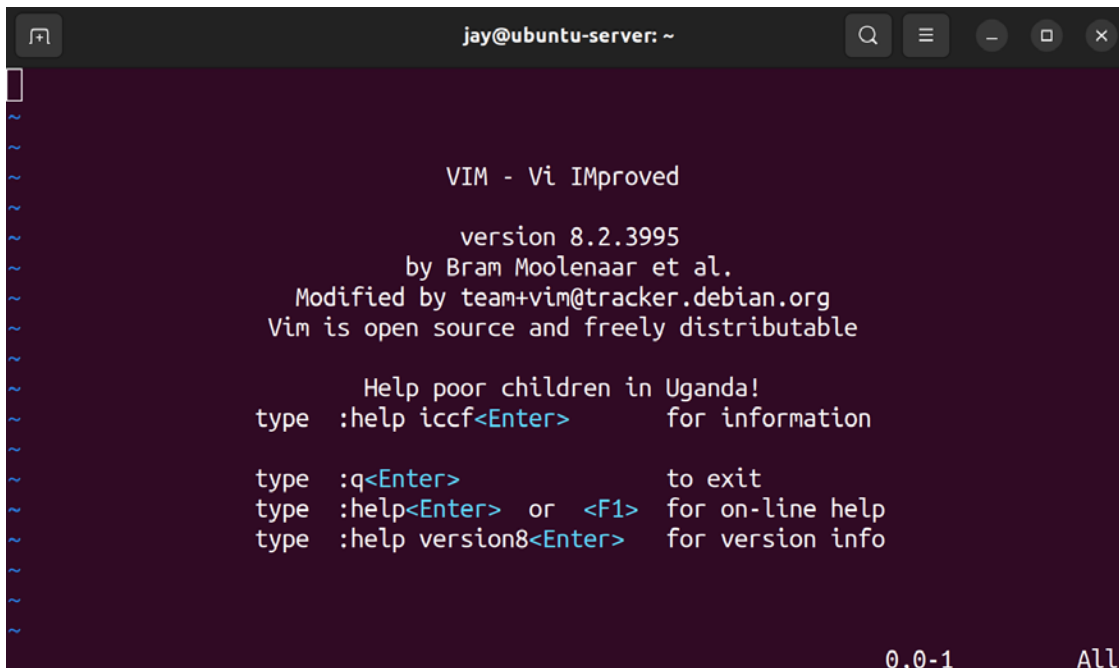
Now, let's take a look at Vim.

Editing with Vim

Vim is my favorite editor, and the one I use most often. It's a little on the advanced side, but it's not to the point where it's frustrating. By default, it's not actually installed. Personally, I prefer the `vim-nox` package, which can be installed by executing the following command:

```
sudo apt install vim-nox
```

It really doesn't matter which of the variations of `vim` you install. Each variation adds its own features. In the case of `vim-nox`, it has built-in support for scripting languages but is not really all that different from standard `vim` in other regards. The concepts you'll learn here are not specific to this version. Just like nano, the `vim` editor can be called by itself by entering the command `vim` with no options, or with a path to a file, such as `vim /home/myuser/myfile.txt`. With no file chosen, `vim` will display its default help text. It gives you some of the default commands, such as `:q` for quitting the editor:

A screenshot of the Vim text editor running in a terminal window. The window title is "jay@ubuntu-server: ~". The terminal background is dark purple. The text displayed is the Vim help screen, which includes the title "VIM - Vi IMproved", the version "8.2.3995", and the author "Bram Moolenaar et al.". It also mentions "Modified by team+vim@tracker.debian.org" and "Vim is open source and freely distributable". There is a section titled "Help poor children in Uganda!" with three entries: "type :help iccf<Enter> for information", "type :q<Enter> to exit", and "type :help<Enter> or <F1> for on-line help". The last entry is "type :help version8<Enter> for version info". At the bottom right, it shows "0,0-1" and "All".

```
jay@ubuntu-server: ~
VIM - Vi IMproved
      version 8.2.3995
      by Bram Moolenaar et al.
 Modified by team+vim@tracker.debian.org
Vim is open source and freely distributable

      Help poor children in Uganda!
type  :help iccf<Enter>      for information
type  :q<Enter>              to exit
type  :help<Enter> or <F1>   for on-line help
type  :help version8<Enter> for version info

0,0-1 All
```

Figure 5.3: The vim text editor, with no file open

When you start `vim`, you start in *command mode*, which is one of several different modes the editor can be in. In command mode, you can't actually edit text. Command mode allows you to run commands, as the name implies, which can allow you to manipulate text in really neat ways.



If you want to edit a file as you were already doing in `nano`, you'll need to switch to *insert mode* to do so. You can switch to insert mode by pressing the *Insert* key on your keyboard, or the *i* letter key. Once you're in insert mode, you can start typing as you normally would in any other editor. You'll move the cursor around with the arrow keys, and insert text wherever you want. Outside of insert mode, you can also navigate with the *h*, *j*, *k*, and *l* keys in place of the arrow keys (which some users actually prefer). To exit insert mode, you can press *Esc*. That will bring you back to command mode.

At first, the different modes of `vim` can be a bit confusing to newcomers. For me, I see it as somewhat of a superpower. With `vim`, you have a mode for editing text and another dedicated mode for manipulating it. When it comes to text manipulation, that's one of the strongest features of `vim`. For an example of this, consider the following hypothetical file:

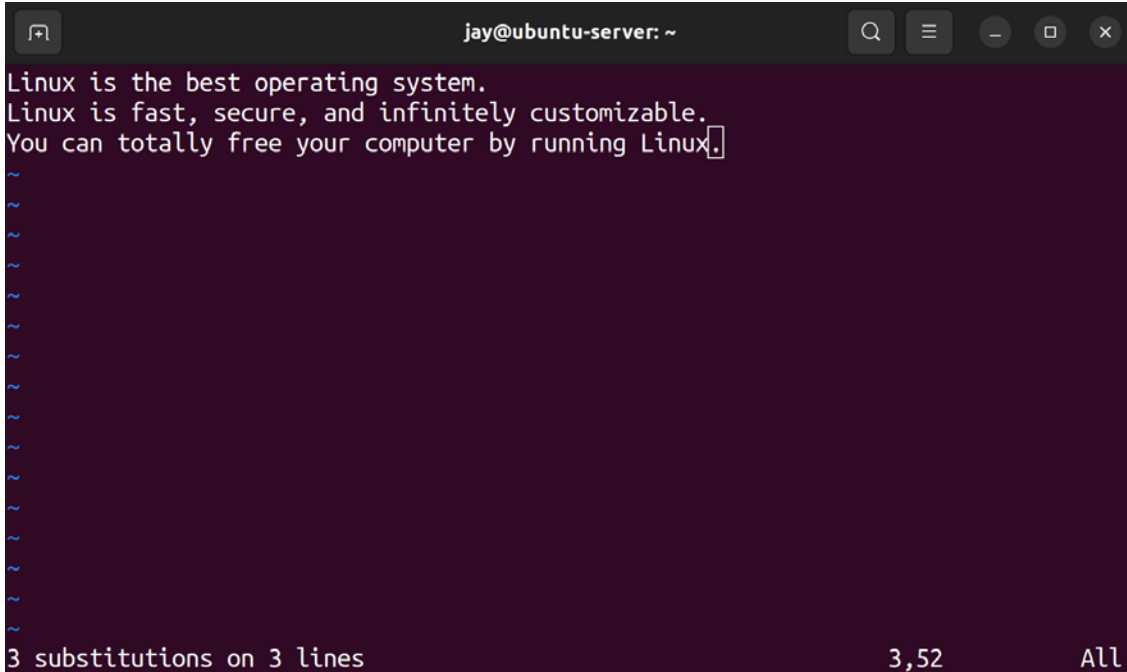
```
jay@ubuntu-server: ~
Windows is the best operating system.
Windows is fast, secure, and infinitely customizable.
You can totally free your computer by running Windows.
~
~
~
~
~
~
~
~
~
~
~
~
~
-- INSERT --                      3,55      All
```

Figure 5.4: The `vim` text editor, with a sample file

I don't know why, but something about that file seems off to me. We'll need to make a very important correction to try to fix it. First, you can see from the screenshot that the editor was left in insert mode. To do some text manipulation, we first need to press *Esc* to go back to command mode. Once there, we can get ready to type a command. And the command we want to type is this one:

```
:%s/Windows/Linux/g
```

I don't know about you, but personally, I think the file looks better now after running that command:



```
linux is the best operating system.  
linux is fast, secure, and infinitely customizable.  
You can totally free your computer by running Linux.  
~  
~  
~  
~  
~  
~  
~  
~  
~  
~  
~  
3 substitutions on 3 lines                               3,52 All
```

Figure 5.5: The vim text editor, after running a sample command

Technically, Linux is a kernel and not an operating system. Ubuntu, a distribution of Linux, is the closest equivalent of an operating system for us. But I'm willing to let that slide, for now. Other than that, in my opinion, this version of the file is more accurate!

This was a more advanced example of vim usage that would normally be outside the scope of this book. I wanted to show you an example of the power of this amazing editor, and finding and replacing text is just one of the many things we can do with it. Some people (including myself) even go as far as to install plugins to essentially turn vim into an **Integrated Development Environment (IDE)**. In the example we've just used, we were able to find and replace all occurrences of a string with another string. This was also an example of the types of tasks we carry out in command mode, which allows us to manipulate text and enter more advanced commands. With this admittedly contrived example, you can immediately see the value of having multiple modes in vim.

I've been using vim for about a decade or so because I can never seem to figure out how to exit it. Okay, to be fair, that joke is getting to be a bit old now. If you've already been using Ubuntu or any other Linux distribution for a while, that joke probably made you cringe a little. If that was the first time you've seen it, then I'm glad I was the one that introduced you to it—for some reason, vim has a reputation for being difficult to exit. But that's not really true at all, it's actually quite easy. In command mode, you simply enter `:q` to exit the editor:

```
:q
```

If you've made changes, the `:q` command won't let you exit, but you can force exit if you add an exclamation mark, so the command becomes `:q!`. If you'd also like to save changes while exiting, add `w`, so the command inside `vim` would become:

```
:wq
```

Essentially, we're exiting the editor (`q`) and writing the file at the same time (`w`). In command mode, commands start with a colon (`:`) and then the actual command. There are so many commands available that it's impossible to talk about them all in one chapter. What I'll do, then, is mention some that I consider to be the most important.

First, it's actually possible to run a shell command without leaving `vim` by using the following command:

```
:! <shell command>
```

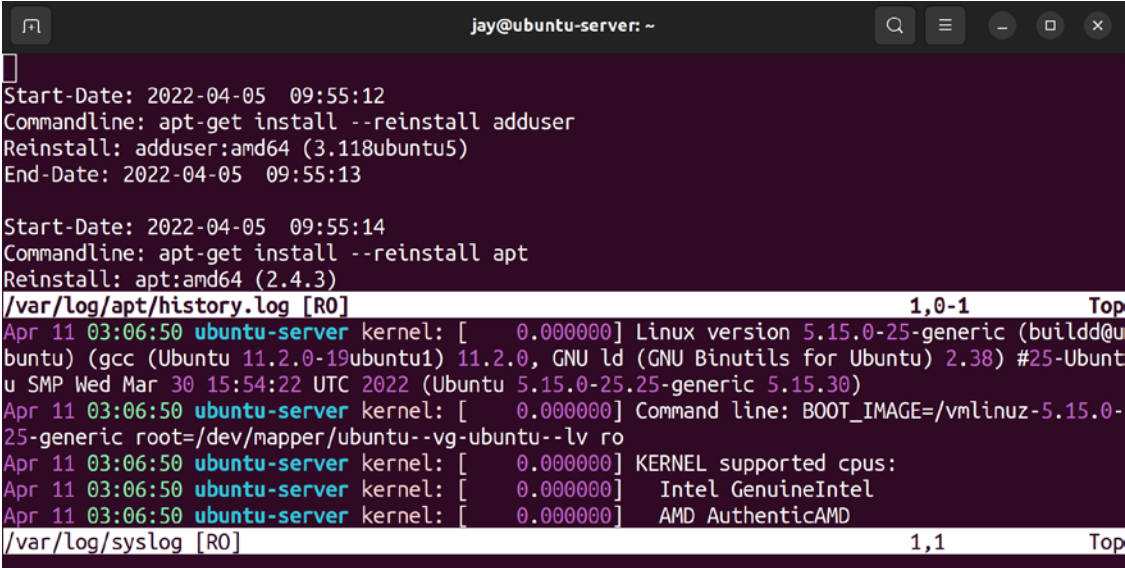
The exclamation mark allows you to run a command, and then you type the actual command. For example:

```
:! ls -l /var/log
```

The previous example will show you the contents of the `/var/log` directory, and then you can press `Enter` to return back to `vim`. The `:sp` command is short for *split*:

```
:sp /path/to/file
```

In this case, `vim` is able to show you more than one file at a time in the same window, effectively splitting the window to show you both files. With `:split`, or `:sp` for short (both do the same thing), it will either split the file into two views (the same file open in each) or it will show a separate file in the other split if you give it a filename. This command will split the file horizontally:



```

jay@ubuntu-server: ~
Start-Date: 2022-04-05 09:55:12
Commandline: apt-get install --reinstall adduser
Reinstall: adduser:amd64 (3.118ubuntu5)
End-Date: 2022-04-05 09:55:13

Start-Date: 2022-04-05 09:55:14
Commandline: apt-get install --reinstall apt
Reinstall: apt:amd64 (2.4.3)
/var/log/apt/history.log [R0] 1,0-1 Top
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Linux version 5.15.0-25-generic (buildd@u
ubuntu) (gcc (Ubuntu 11.2.0-19ubuntu1) 11.2.0, GNU ld (GNU Binutils for Ubuntu) 2.38) #25-Ubunt
u SMP Wed Mar 30 15:54:22 UTC 2022 (Ubuntu 5.15.0-25.25-generic 5.15.30)
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Command line: BOOT_IMAGE=/vmlinuz-5.15.0-
25-generic root=/dev/mapper/ubuntu--vg-ubuntu--lv ro
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] KERNEL supported cpus:
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Intel GenuineIntel
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] AMD AuthenticAMD
/var/log/syslog [R0] 1,1 Top

```

Figure 5.6: The `vim` text editor, with two files open in the same window

The previous screenshot shows two files open, `/var/log/apt/history.log` and `/var/log/syslog`. You may also notice that the status bar for each file, at the bottom, shows `RO`. As you can probably guess, this stands for *read only* and displays on the screen because I'm viewing files that only root has access to change, and I didn't use `sudo`.

To switch between the two files, we can press the `w` key twice while holding `Ctrl`. When we do that, our insertion point moves from one split to the other. To exit each individual buffer (or the editor itself, if we only have one file open), we can press `Esc` to return to command mode, and then enter the `:q` or `:q!` command to exit without saving changes, or `:wq` to exit while also saving changes, just like before.

An alternative to `:split` is `:vsplit`, or `:vs` for short. It does the same thing as `:split`, but it splits the window vertically. Considering most computer displays nowadays are widescreen (and even ultra-widescreen), a vertical split often works out better in a practical sense:

```

jay@ubuntu-server: ~
Start-Date: 2022-04-05 09:55:12
Commandline: apt-get install --reinstall adduser
Reinstall: adduser:amd64 (3.118ubuntu5)
End-Date: 2022-04-05 09:55:13

Start-Date: 2022-04-05 09:55:14
Commandline: apt-get install --reinstall apt
Reinstall: apt:amd64 (2.4.3)
End-Date: 2022-04-05 09:55:15

Start-Date: 2022-04-05 09:55:16
Commandline: apt-get install --reinstall base-files
Reinstall: base-files:amd64 (12ubuntu3)
End-Date: 2022-04-05 09:55:16

Start-Date: 2022-04-05 09:55:17
Commandline: apt-get install --reinstall base-passwd
Reinstall: base-passwd:amd64 (3.5.2build1)
End-Date: 2022-04-05 09:55:18

Start-Date: 2022-04-05 09:55:19
Commandline: apt-get install --reinstall bash
/var/log/apt/history.log [RO] 1,0-1 Top /var/log/syslog [RO] 1,1 Top
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Li
nux version 5.15.0-25-generic (buildd@ubuntu) (gcc (Ubu
ntu 11.2.0-19ubuntu1) 11.2.0, GNU ld (GNU Binutils for
Ubuntu) 2.38) #25-Ubuntu SMP Wed Mar 30 15:54:22 UTC 20
22 (Ubuntu 5.15.0-25.25-generic 5.15.30)
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] Co
mmand line: BOOT_IMAGE=/vmlinuz-5.15.0-25-generic root=
/dev/mapper/ubuntu--vg-ubuntu--lv ro
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] KE
RNL supported cpus:
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000]
Intel GenuineIntel
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000]
AMD AuthenticAMD
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000]
Hygon HygonGenuine
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000]
Centaur CentaurHauls
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000]
zhaoxin Shanghai
Apr 11 03:06:50 ubuntu-server kernel: [ 0.000000] x8
6/fpu: x87 FPU will use FXSAVE
@@@

```

Figure 5.7: The vim text editor, with two files open in a vertical split

To be honest, the effect doesn't look all that great in the previous screenshot, the reason being the width of a page in a book isn't enough to really show the benefit. Go ahead and give it a shot and see for yourself.

So far, we've gone over two modes of vim—command and insert mode. But there's also another commonly used mode we haven't discussed yet—*visual mode*. Visual mode allows you to select text, which then allows you to perform tasks such as copy and paste.

To do so, make sure you're in command mode, and then start by moving the cursor to the first character in a series of text you want to copy. Then, press the `v` letter key on your keyboard, and move the selection with your arrow keys.

You'll notice that you'll highlight more text as you move the cursor. Once you've selected all the text that you'd like to copy, press the *y* key and the highlight will go away. At this point, you've essentially copied the text you had highlighted into the *vim* equivalent of the clipboard, similar to desktop operating systems. You can then (while also in command mode) press the *p* key to paste the text wherever the cursor is. It may take a few tries to get a hang of the workflow, but go ahead and give it a shot. If you make a mistake, you can press *u* while in command mode to *undo*, so you can revert any changes you make.

Again, *vim* is a somewhat advanced editor. Learning the basic functions is fairly easy, so knowing how to open, edit, close, and save files is something you can learn fairly quickly. The feature set of *vim* is so vast that I'm still learning new tricks many years after I started using it. To make sure you understand the basics though, let's summarize the basic workflow of *vim* to help you memorize it. It looks something like this:

1. Open *Vim* by either typing *vim* by itself, or with a filename: `vim <filename>`.
2. You start in *command mode*. This mode is great for running *vim* commands. Press the *Insert* or the *i* key to switch to *insert mode*.
3. While in insert mode, you can edit the text by typing with your keyboard and moving the cursor around with the arrow keys.
4. When you're done editing the file, press *Esc* to return to command mode.
5. To exit without saving changes, enter the `:q` or `:q!` command. If you do want to save changes, enter `:w` by itself to save and remain in the editor, or `:wq` to save and quit *vim* at the same time.

Entire books could be written about *vim* (and there have been), so it would be outside the scope of this book to go into exhaustive detail. In fact, I even have a dedicated video tutorial series about it at www.learnlinux.tv if you're interested in learning more. But for now, I'll leave you with a few more useful tips that I think you should know.

While in command mode, you can press the *x* key to delete a single character, wherever the cursor is. You can press the *d* letter key *twice* in succession to delete an entire line. When you do this, that line is also copied into the paste buffer; you can paste that line by pressing *p*.

In addition, I mentioned several times that you can switch to insert mode by pressing *i* or *Insert*. You can also do so by pressing the *a* letter key with or without holding *Shift*. If you don't hold *Shift*, you'll enter insert mode one character to the right. This isn't very useful to me, but typing a capital *A* is something I find myself doing a lot, which enters you into insert mode while also taking you to the very end of the current line. Since I often want to start typing after the end of a sentence, this works well for me.

Also, while in command mode, you can type *Shift* + *g* to immediately go to the very end of the file. Alternatively, you can press the *g* key *twice* in succession to move to the very top of the file.

Another trick I like is to turn on line numbers. This is useful, especially if an error message in a log file is complaining about a file regarding a specific line. The following command turns on line numbers:

```
:set number
```

The following command disables line numbers:

```
:set nonumber
```

If you'd always like to see line numbers by default, you can edit the `.vimrc` file inside your home directory. This file is read by `vim` each time you start it.

Note that the `.vimrc` file begins with a period, which means that the file is hidden. Hidden files aren't normally shown to you when you use `ls` to list the contents of a directory. Using the `-a` option with the `ls` command will show all files, including the ones that are normally hidden.

Go ahead and open the `.vimrc` file in your editor:

```
vim ~/.vimrc
```

Most likely, this will be a blank file for you as a `.vimrc` file doesn't exist by default. There are many commands we can add to tweak the behavior of the `vim` editor, too many to mention in a single chapter. The following line is very useful though, as it ensures that line numbers are enabled by default:

```
set number
```

Leaving out the colon at the beginning of the line is intentional; it's not required inside the `.vimrc` file. From this point forward, each new `vim` session you open will start off with line numbers enabled. You can still disable line numbers as needed with the `:set nonumber` command within `vim` as we did earlier. In addition, you can add many more customizations to `vim` with the `.vimrc` file, but there are simply too many possibilities to mention here. For now, just know that customizing `vim` with the config file is something that you can do.

Now that we know how to edit files, we should also take a closer look at streams, which allow us to manipulate both input and output in various ways.

Input and output streams

During our journey into Ubuntu Server so far, we've worked quite a bit within the terminal. We've been able to inspect the contents of files, insert text into files, and more. We've actually been working with **streams** the entire time without knowing it. In this section, we're going to talk about this subject in more detail.

If you've studied computer science at all, then you probably already know that *output* refers to things that are printed out of the computer (for example, text being printed to the screen, or onto paper from a printer) and *input* refers to data that is being entered into a computer, whether that be on the command line, into a file, or similar.

Linux takes this concept a bit further. Streams in Linux refer to a special way to handle what's going in or out, and beyond the input and output streams, we also have a third that refers to errors.

Output streams in Linux are referred to as **Standard Output**, and input streams are referred to as **Standard Input**. These are abbreviated as **stdout** and **stdin**, respectively. The reason for the expansion of the simple concept of input/output into a concept of its own is because, on the Linux shell, we can handle these streams differently and perform different tasks with them.

We've been working with standard output throughout the entire book so far. Everything that is printed to the terminal is standard output. For example, when you ran the `sudo apt install vim-nox` command earlier, the results of that command (the flood of text showing the status of the package installation as it occurred) was standard output. When you use the `cat /var/log/syslog` command to have the contents of `/var/log/syslog` dumped onto the screen, the contents being displayed are standard output. By far, standard output is something you'll work with the most.

To understand the concept of standard output better, let's take a look at redirection. The following command is an example:

```
cat /var/log/syslog > ~/logfile.txt
```

With that command, we're using `cat` to display the contents of `/var/log/syslog`. But instead of simply showing the contents onscreen we use the `>` character to redirect standard output into a file, `~/logfile.txt`. This means that standard output (showing the contents of the file) will not be shown at all, since we redirected that into a file. Similarly, we could have also run:

```
cat /var/log/syslog 1> ~/logfile.txt
```

Notice that I added a `1` right before the *greater-than* symbol. Standard output is designated by a file descriptor of `1`. So, with that command, I'm specifically saying that I want to redirect standard output, and *only* standard output, into the file. Standard output is implied, so I didn't need to include the `1`. That's why simply using the *greater-than* symbol works for redirecting standard output to a file.

If you want to append a file rather than completely overwrite it, you can use two *greater-than* symbols (`>>`) to append, rather than overwrite. For example, the following command will add the contents of the `syslog` file to the end of the `logfile.txt` file, rather than overwrite the entire file:

```
cat /var/log/syslog >> ~/logfile.txt
```

Standard input also has a file descriptor, which is `0`. Standard input is how commands receive data. Essentially, commands that accept input from the user do so by accepting `stdin`. Standard input is a bit more challenging to show in an example, but the following works:

```
cat /var/log/syslog | grep -i <keyword>
```

With that command, I'm grabbing the contents of the `/var/log/syslog` file and piping that into the `grep` command, which allows me to display only the lines that include a specific term. No two `syslog` files will be the same, since every server can be configured uniquely. But if you're attempting to investigate a specific application or service, you can `grep` the log for keywords you think might be appropriate. If you include the `-i` option, then `grep` will perform a search that's case-insensitive. In this example, the output of the `cat` command becomes standard input to the `grep` command. This command can be executed with any search term of your choice.

I could have also run:

```
cat < /var/log/syslog | grep Network
```


In that example, I'm using the *less-than* symbol to redirect the contents of `/var/log/syslog` to be standard input to the `cat` command. The `cat` command normally prints whatever text is fed to it to the screen as standard output, but instead, I'm using the pipe symbol, `|`, to grab that output and use it as standard input to the `grep` command.

This concept can be a bit confusing at first, but if you keep practicing it, it will definitely make sense. Let's look at another example, so we can understand standard error (`stderr`) as well:

```
find / -name "syslog"
```

The `find` command allows you to find files that match particular criteria, such as looking for files that are named `syslog` in this example. Here, we're searching the entire filesystem, because we used `/` to start the search from. The problem is I don't have permission to read all the files on the filesystem, and I didn't use `sudo`. This is going to result in quite a few errors being printed to the screen, errors such as these:

```
find: '/var/lib/netdata/health': Permission denied
find: '/var/lib/netdata/registry': Permission denied
find: '/var/lib/netdata/cloud.d': Permission denied
find: '/var/lib/udisks2': Permission denied
```

Since the `find` command was used to search the entire filesystem, including places I don't have permission to look, our terminal will be flooded with errors. These errors are displayed using standard error, which has a file descriptor of 2. If we want to hide these errors, we can do this:

```
find / -name "syslog" 2> /dev/null
```

With that command, no errors will show up when we run it. That's because we instructed the interpreter to capture standard error and redirect it to `/dev/null`. `/dev/null` is a special device where things disappear forever. If you move or redirect something there, it's effectively deleted. Since a standard error has a file descriptor of 2, we combined that with the *greater-than* symbol to form `2>`, which basically instructs the shell to do a redirect, but to only redirect standard error and leave standard output alone. We can also choose to redirect more than one stream to different places within a single command:

```
find / -name "syslog" 1> stdout.txt 2> stderr.txt
```

With that variation, I'm redirecting successful output to `stdout.txt`, and the errors to `stderr.txt`. This allows us full control over where successful and unsuccessful messages are printed. This also helps us quite a bit when it comes to troubleshooting, because we may want to focus on only the errors, and getting rid of the success output may trim the number of lines we'll have to look through.

I recommend that you go ahead and practice this concept as much as you can; it's definitely something you'll want to commit to memory. You don't have to master this concept right now, but knowing the basics will provide a great foundation.

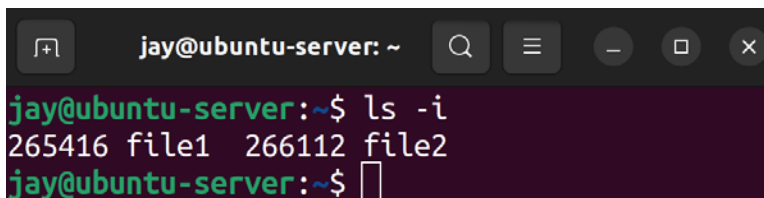
Next, let's discuss another aspect of file management—links. Sometimes, you'll need to link one thing to another, and there are a few ways to go about that along with some best-practice advice.

Using symbolic and hard links

If you've used a graphical operating system for more than a week, you're probably more than familiar with the concept of shortcuts. Either on the desktop or within a menu, you will have shortcuts to files and applications. This could be a shortcut to your home or profile directory, a shortcut to an application, an individual file, and so on. We have the same concept in Linux.

With Linux, we can link files to other files, which gives us the ability to create our own shortcuts, which are effectively similar to shortcuts in graphical operating systems, but without the requirement of a GUI. This comes in the form of *symbolic and hard* links, which are two different methods by which we can link things. Symbolic and hard links are very similar, but to explain them, you'll first need to understand the concept of **inodes**.

An inode is a data object that contains metadata regarding files within your filesystem. Although a full walkthrough of the concept of inodes would be quite long, think of an inode as a type of database object, containing metadata for the actual items you're storing on your disk. Information stored in inodes includes details such as the owner of the file, permissions, last modified date, and type (whether it is a directory or a file). Inodes are represented by an integer number, which you can view with the `-i` option of the `ls` command. On my system, I created two files: `file1` and `file2`. These files are inodes 265416 and 266112, respectively. You can see this output in the following screenshot where I run the `ls -i` command. This information will come in handy shortly:



```
jay@ubuntu-server: ~  
jay@ubuntu-server:~$ ls -i  
265416 file1 266112 file2  
jay@ubuntu-server:~$
```

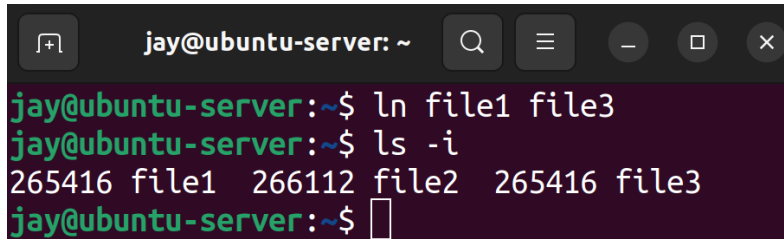
Figure 5.8: Output of `ls -i`

As I mentioned earlier, there are two types of links in Linux: the symbolic link and hard link. While the two types of links approach the concept differently, they pretty much serve the same purpose. Basically, a link allows us to reference a file somewhere else on our filesystem.

For a practical example, let's create a hard link. In my case, I have a couple of files in a `test` directory, so I can create a link to either of them. To create a link, we'll use the `ln` command:

```
ln file1 file3
```

Here, I'm creating a hard link (`file3`) that is linked to `file1`. To play around with this, go ahead and create a link to a file on your system. If we use `ls` again with the `-i` option, we'll see something interesting:



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ ln file1 file3
jay@ubuntu-server:~$ ls -i
265416 file1 266112 file2 265416 file3
jay@ubuntu-server:~$

```

Figure 5.9: Output of second `ls -l` command

If you look closely at the output, both `file1` and `file3` have the same inode number. Essentially, a hard link is a duplicate directory entry, where both entries point to the same data. In this case, we created a hard link that points to another file. With this hard link created, we can move `file3` into another location on the filesystem and it will still be a link to `file1`. Hard links have a couple of limitations, however. First, you cannot create a hard link to a directory, only a file. Second, this link cannot be moved to a different filesystem. That makes sense, considering each filesystem has its own inodes. Inode 265416 on my system would, of course, not point to the same file on another system, if this inode number even exists at all.

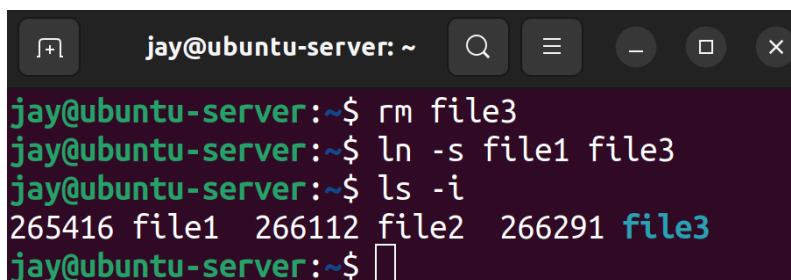
To overcome these limitations, we can consider using a soft link instead. A soft link (also known as a **symbolic link** or **symlink**) is an entry that points to another directory or file. This is different from a hard link because a hard link is a duplicate entry that references an inode, while a symbolic link references a specific path. Symbolic links can not only be moved around between filesystems; we can also create a symbolic link to a directory as well. To illustrate how a symbolic link works, let's create one. In my case, I'll delete `file3` and recreate it as a symbolic link. We'll again use the `ln` command:

```

rm file3
ln -s file1 file3

```

With the `-s` option of `ln`, I'm creating a symbolic link. First, I deleted the original hard link with the `rm` command (which doesn't disturb the original file, `file1`) and then created a symbolic link, also named `file3`. If we use `ls -i` again, we'll see that `file3` does not have the same inode number as `file1`:



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ rm file3
jay@ubuntu-server:~$ ln -s file1 file3
jay@ubuntu-server:~$ ls -i
265416 file1 266112 file2 266291 file3
jay@ubuntu-server:~$

```

Figure 5.10: Output of `ls -i` after creating a symbolic link

Notice that the inode numbers of each file are all different. At this point, the main difference compared with a hard link should become apparent. A symbolic link is not a clone of the original file; it's simply a pointer to the original file's path. Any commands you execute against `file3` are actually being run against the target that the link is pointing to. Hard links, on the other hand, point directly to the file.

In practice, symbolic links are incredibly useful when it comes to server administration. However, it's important not to go crazy and create a great number of symbolic links all over the filesystem. This certainly won't be a problem for you if you are the only administrator on the server, but if you resign and someone takes your place, it will be a headache for them to figure out all of your symbolic links and map where they lead to. You can certainly create documentation for your symbolic links, but then you'd have to keep track of them and constantly update documentation. My recommendation is to only create symbolic links when there are no other options, or if doing so benefits your organization and streamlines your file layout.

Getting back to the subject of symbolic links versus hard links, you're probably wondering which you should use and when to use them. The main benefit of a hard link is that you can move either file (the link or the original) to anywhere on the same filesystem and the link will not break. This is not true of symbolic links; however, if you move the original file, the symbolic link will be pointing to a file that no longer exists at that location. Hard links are basically duplicate entries pointing to the same object, and thus have the same inode number, so both will have the same file size and content. A symbolic link is a pointer to the file's path—nothing more, nothing less.

However, even though I just spoke about the several benefits of hard links, I actually recommend symbolic links for most use cases. They can cross filesystems, can be linked to directories, and it's easier to determine from the output where they lead. If you move hard links around, you may forget where they were originally located or which file actually points to which. Sure, with a few commands you can find them and map them easily. But overall, symbolic links are more convenient in the long run. As long as you're mindful of recreating your symbolic link whenever you move the original file (and you use them only when you need to), you shouldn't have an issue.

Summary

In this chapter, we elevated our terminal kung-fu to another level and looked at concepts such as moving and copying files. We continued into a discussion of two popular text editors, Nano and Vim. Then, we took a dive into the subject of streams, and finished off the chapter with an understanding of the differences between symbolic and hard links, as well as how to create them.

In *Chapter 6, Boosting Your Command-line Efficiency*, we'll dive a bit deeper into command-line tips and tricks, which will include a discussion on Bash history, writing basic scripts, and more.

Relevant videos

- Linux Crash Course – Data Streams (LearnLinuxTV): <https://linux.video/se-data-streams>
- Linux Crash Course – nano (LearnLinuxTV): <https://linux.video/le-nano>
- Linux Crash Course – Symbolic Links (LearnLinuxTV): <https://linux.video/se-symlinks>
- Vim Text Editor Tutorial Series: <https://learnlinux.link/vim>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



6

Boosting Your Command-line Efficiency

Throughout this book, we've been using the command line quite heavily. Using the shell, we've installed packages, created users, edited configuration files, and more. In the last chapter, we took a look at file management to enhance our terminal skills further. This time around, we'll dedicate an entire chapter to the shell, with the goal of becoming more efficient with it. Here, we'll take what we already know and add some useful time-saving tips, some information on looping, variables, and we'll even look into writing scripts.

In this chapter, we will cover the following topics:

- Understanding the Linux shell
- Understanding Bash history
- Learning some useful command-line tricks
- Understanding variables
- Writing simple scripts
- Putting it all together: Writing an rsync backup script

Let's begin this chapter with further discussion regarding the Linux shell, which will help us better understand how we're interacting with the server while we enter commands.

Understanding the Linux shell

When it comes to the Linux shell, it's important to understand what exactly the term pertains to. We've been using the command line repeatedly throughout the book, but we haven't yet had any formal discussion about the actual interface through which our commands are entered.

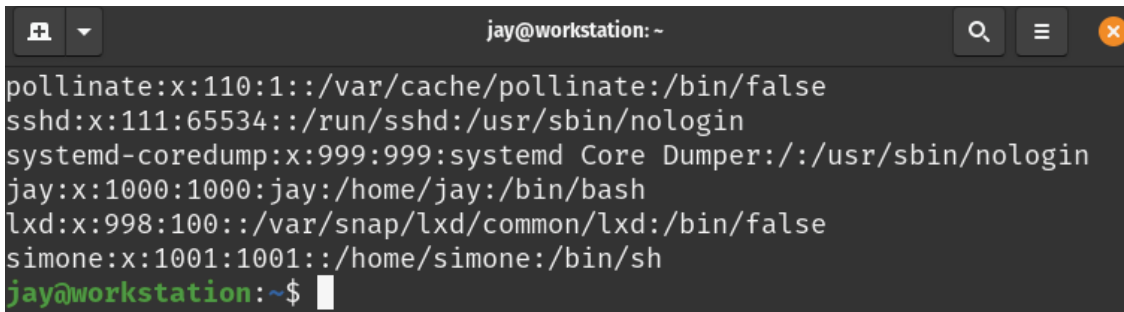
Essentially, we've been entering our commands into a command interpreter known as the **Bourne Again Shell**, or simply **Bash**. Bash is just one of many different *shells* that you can use to enter commands.

There are other options, including **Zsh**, **Fish**, and **ksh**, but **Bash** is the default command shell for the majority of Linux distributions. It's even available on macOS (although the default on that platform is Zsh nowadays), as well as on Windows by installing the Windows Subsystem for Linux. Therefore, by understanding the basics of Bash, your knowledge will be compatible with other distributions and platforms. While it's fun to learn other shells such as Zsh, Bash is definitely the one to focus the most attention on if you're just starting.

You may wonder, then, where you configure the shell that your user account will use. If you recall from *Chapter 2, Managing Users and Permissions*, we looked at the `/etc/passwd` file. As I'm sure you remember, this file keeps a list of user accounts available on the system. Go ahead and take a look at this file to refresh yourself by entering the following command:

```
cat /etc/passwd
```

This will produce an output like the one shown in *Figure 6.1*:



```
pollinate:x:110:1::/var/cache/pollinate:/bin/false
sshd:x:111:65534::/run/sshd:/usr/sbin/nologin
systemd-coredump:x:999:999:systemd Core Dumper::/usr/sbin/nologin
jay:x:1000:1000:jay:/home/jay:/bin/bash
lxd:x:998:100::/var/snap/lxd/common/lxd:/bin/false
simone:x:1001:1001:/home/simone:/bin/sh
jay@workstation:~$
```

Figure 6.1: The last several lines of a sample `/etc/passwd` file

See the last field in every entry? That's where we configure which shell is launched when a user logs in or starts a new terminal session. Unless you've already changed it, the entry for your user account should read `/bin/bash`.

You'll see other variations in this file, such as `/bin/false` or `/usr/sbin/nologin`. These are actually invalid shells that will prevent a user from logging in to the system if one of these is set as the default for that user. While it may seem strange to set a user's shell to one that prevents them from logging in, this practice is fairly common – not all user accounts need to log in to a server.

System accounts also exist alongside normal user accounts, which are created for background jobs. These system users won't need to actually log in to the system in order to do their work, so it's common to take that a step further and set the shell of a system user to an invalid shell, so that way, the account cannot be used to log in to the server if it's taken over by an outside threat actor (the less an account can do, the safer it is).

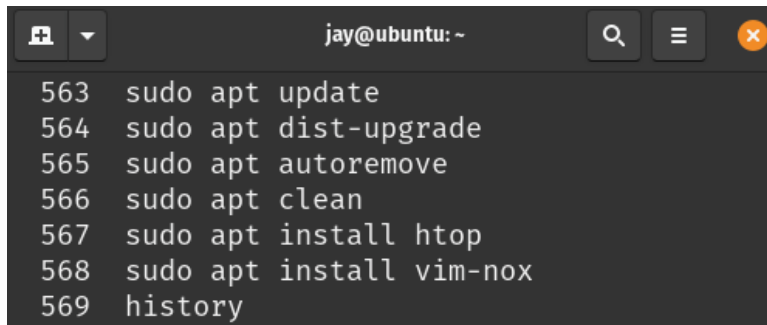
The shell program itself is responsible for reading the commands you type and having the Linux kernel execute them. Some shells, Bash notably, have additional features, such as *history*, that are very useful to administrators.

Understanding Bash history

Speaking of history, let's dive right into that concept. By default, Bash keeps track of all the commands you enter during your sessions, so that if you need to recall a previously entered command, you can definitely do so. History also serves another purpose, and that is seeing what other users have been up to. However, since users can edit their own history to cover their tracks, it's not always useful for that purpose.

You may have already seen Bash's history feature in some form, if you've ever pressed the up and down arrows on the shell to recall a previously used command. If you didn't already know you could do that, go ahead and give it a try. You should see that by pressing the up and down arrows, you can cycle through commands that you've used previously.

Another trick is that you can also simply type `history` in the shell and see a list of previously entered commands, as shown in *Figure 6.2*:

A terminal window titled 'jay@ubuntu: ~' showing the output of the 'history' command. The output is a list of commands with their corresponding history numbers: 563 sudo apt update, 564 sudo apt dist-upgrade, 565 sudo apt autoremove, 566 sudo apt clean, 567 sudo apt install htop, 568 sudo apt install vim-nox, and 569 history.

```
563 sudo apt update
564 sudo apt dist-upgrade
565 sudo apt autoremove
566 sudo apt clean
567 sudo apt install htop
568 sudo apt install vim-nox
569 history
```

Figure 6.2: Output from the history command

At this point, you can copy and paste a command you've used previously from this list to run it again. In fact, there's an even easier way. Do you notice the number on the left of each command? We can utilize that number to quickly recall a previously used command. In my screenshot, item 566 is where I ran `sudo apt update`. If I wanted to run that same command again, I could simply enter the following command:

```
!566
```

In this case, I typed just four characters, and I was able to recall the previously used command, which performs the same action as typing this:

```
sudo apt update
```

That saves a lot of typing, which is great because we administrators want to type as little as possible (unless we're writing a book).

Let's look at a few additional history commands we can use. First, if we want to delete something from the history, we can simply do this:

```
history -d 566
```


In this example, we deleted item 563 from Bash's history. To delete a different history entry, simply replace 563 with whatever the number is for the item we would want to remove. You may be wondering why deleting something from the history would be necessary. The answer to that is simple: sometimes we make mistakes. Perhaps we mistyped something, and we don't want a junior administrator to look at the history and rerun an invalid command. Worse, if we accidentally saved a password to the history, it would be there for all to see. We would definitely want to remove that item so that the password isn't saved in plain text in the history file. One very common example of this is with MySQL or MariaDB. When you enter the MySQL or MariaDB shell, you can use the `-p` option and type the password in one line. It would look something like this:

```
mariadb -u root -pSuperSecretPassword
```

That command may appear useful, because in one command you'd be logged in to your database server as its own version of the `root` user. However, this is one of my pet peeves—I really don't like it when people run commands that include a password in the clear. Having the `root` password in your shell's history is a HUGE security risk. This is just one example of something you won't want in our Bash history, though. My main goal here is to demonstrate that you should think about security when entering commands. If you have a potentially sensitive item in your command history, you should remove it. In fact, you can actually enter a command and not have it saved in the history at all. Simply prefix the command with a space. If you do, it will not be recorded in the history file. Go ahead, give it a try and see for yourself.

Having commands prefixed with a space ignored in Bash is actually a custom option enabled by default in Ubuntu Server. Not all distributions include this feature enabled by default. If you're using a distribution that doesn't have this enabled already, add the following to your `.bashrc` file (we will talk about this file in greater detail later).

```
HISTCONTROL=ignoreboth
```

This configuration line also causes duplicate commands to not be entered into the history file as well, which can condense the history file.

So, you might be wondering, where is this history information actually stored? Check out the `.bash_history` file, which is found in your home directory (or `/root` for the `root` user). When you exit your shell, your history is copied to that file. If you remove that file, you're effectively clearing your history. I don't recommend you make a habit of that, though. Having a history of commands is very useful, especially when you may not remember how you solved a problem last time. History in Bash can save you from looking up a command again. To find out more about what the history command can do, check out the relevant man page with `man history`.

Learning new tricks with the command line that allow me to work more efficiently is a great feeling, at least for me. In the next section, we'll explore some useful tricks we can utilize while working with the shell.

Learning some useful command-line tricks

Productivity hacks utilizing the shell are some of my favorite things in this world, right up there with music, video games, and Diet Pepsi. There's nothing like the feeling you get when you discover a useful feature that saves you time or increases your efficiency. There are many things I've discovered along the way that I wish I had known earlier on. One of my goals while writing this book is to teach you as many things as I can that took me longer to learn than I'm comfortable admitting to. In this section, in no particular order, I'll go over a few tricks that increased my workflow.

First, entering `!!` (two exclamation marks) in your terminal will repeat the command you last used. By itself, this may not seem like much. After all, you can press the up arrow key once and press *Enter* to recall the previous command and execute it. But, when paired with `sudo`, `!!` becomes more interesting. Imagine for a moment that you entered a command that needs root privileges, but you forgot to use `sudo`. We've all made this mistake. In fact, as of the time I'm writing this chapter, I've been using Linux for 20 years and I *still* forget to use `sudo` from time to time. When we forget `sudo`, we have to type the command all over again. Or, we can just do this:

```
sudo !!
```

And just like that, you prefixed the previously used command with `sudo` without having to completely retype it.

Speaking of avoiding unnecessary typing, a very easy (yet incredibly useful) feature is **Tab Completion**. Often, the Bash shell will be able to automatically complete part of your commands. If you start typing a few characters of a command or path, press *Tab* on your keyboard, and if the characters you've typed are enough to narrow down the result, the shell will complete the path for you. You can also press *Tab* twice in succession to see a list of possibilities that match the characters you've typed so far. Go ahead and give it a shot. By way of a quick example, you can type `ls` along with the path to your home directory, leaving some characters out on purpose. Then press *Tab*, and see whether the command automatically completes. For example, I can type the following into the terminal:

```
ls /home/j
```

And after I press *Tab*, it completes the command for me:

```
ls /home/jay
```

In addition, there are other special keyboard keys that will help you to navigate the command line quicker. Here's a table containing some of the most useful keyboard shortcuts:

Keyboard shortcut	Result
<i>Ctrl + a</i>	Moves the cursor to the beginning of the line
<i>Ctrl + e</i>	Moves the cursor to the end of the line
<i>Ctrl + l</i>	Clears the screen
<i>Ctrl + k</i>	Deletes characters from the cursor to the end of the line
<i>Ctrl + u</i>	Deletes everything you've typed on that line (also works to clear text while entering a password)
<i>Ctrl + w</i>	Deletes the word to the left of the cursor

Going a bit further into the command history, we can also press *Ctrl + r* on the shell to initiate a search. After pressing these keys, we can start typing a command, and we'll get a preview of a command that matches what we're typing, which will be narrowed down further as we type more characters of it. This is one of those things that is hard for me to describe, and screenshots certainly don't help here, so go ahead and just give it a shot. For example, press *Ctrl + r* and then start typing `sudo apt`. The last time you used that command should appear, and you can press *Ctrl + r* again, and again, and again to see additional examples of commands that you've typed in the past that contain those characters. When you get efficient with this, it's actually faster than the `history` command, but it takes a bit to get used to this.

Another fun trick is editing a command you've previously typed in a text editor. I know this sounds strange, but hear me out. Assume you pressed the up arrow, you have a very long command, and you just want to edit part of it without having to execute the entire thing, perhaps a command like this:

```
sudo apt update && sudo apt install apache2
```

Let's assume you want to install `nginx` instead of `apache2`, but the rest of the command is right. If you hold *Ctrl* and then press *x* followed by *e*, the command will open in a text editor. There, you can change the command. Once you're done making your changes, the command will execute once you save the file. Admittedly, this is usually only useful when you have a very long command and you need to just change part of it. It's also a little weird, but so are computers.

Did you notice the two `&` symbols in the previous command? This is another useful trick; you can actually chain commands together. In the previous example command, we're telling the shell to execute `sudo apt update`. Next, we're telling the shell to then execute `sudo apt install apache2`. The double ampersand is known as the logical AND operator, so the second command is run if the first was successful. If the first command was successful, the second command will execute right after. Another method to chain commands is this:

```
sudo apt update; sudo apt install apache2
```

The difference with the semicolon is that we're telling the shell to execute the second command *regardless* of whether the first command was successful. You may then be wondering, what constitutes success on the shell? An obvious answer to this question might be "it's successful if there are no error messages." While that's often true, the shell utilizes exit codes to programmatically attribute success or failure. You can see the exit code of a command by typing this immediately after the previous command finishes:

```
echo $?
```

An exit code of 0 means success; anything else is some sort of error. Different programs will attribute different codes to different types of failures, but 0 is always a success. With this command, what we're actually doing is printing the content of a variable. `?` is actually a variable, which in this case only exists to hold an exit code. The `echo` command itself can be used to print text to the shell, but it's often used to print the contents of a variable (we'll get into this in more detail in the *Understanding variables* section).

Now, it's time for my favorite time-saving trick of them all—command aliases. The concept of an alias is simple: it allows you to create a command that is just another name for another command. This allows you to simplify commands down to just one word or a few letters. Consider this command, for example:

```
alias install="sudo apt install"
```

When you enter the previous command, you will receive no actual output. But what happens is now you have a new command available—`install`. This command isn't normally available; you just created it with this command.

You can verify that the alias was created successfully by simply running the `alias` command, which will show you a list of aliases present in the shell. If you create a new alias, you should see it in the output. You'll also see additional aliases in the output that you did not create. This is because Ubuntu sets up some by default. In fact, even the `ls` command is an alias!

With this new alias created, any time you execute `install` on the command line, you're instead executing `sudo apt install`. Now, installing packages becomes simpler:

```
install tmux
```

Just like that, you installed `tmux`. You didn't have to type `sudo apt install tmux`; you just simplified the first three words in the command into `install`. In fact, you can simplify it even further:

```
alias i='sudo apt install'
```

Now, you can install a package with this:

```
i tmux
```

With aliases, you can get very creative. Here are some of my personal favorites.

View the top 10 most-CPU-consuming processes:

```
alias cpu10='ps -L aux | sort -nr -k 3 | head -10'
```

View the top 10 most-RAM-consuming processes:

```
alias mem10='ps -L aux | sort -nr -k 4 | head -10'
```

View all mounted filesystems, and present the information in a clean tabbed layout:

```
alias lsmount='mount | column -t'
```

Clear the screen by simply typing `c`:

```
alias c=clear
```

What other aliases can you come up with? Think of a command you may use on a regular basis and simplify it.

There's one issue though, and that is the fact that when you exit your terminal window, your aliases are wiped out. How do you retain them? That leads me to my next productivity trick, editing your `.bashrc` file. This file is present in your home directory and is read every time you start a new terminal session. You can add all of your `alias` commands there; just add them somewhere in the file (for example, at the end). You will need to include the entire command, beginning with `alias` and ending with the commands in quotes. If you wanted to steal my example aliases, you would enter the following lines somewhere in your `.bashrc` file:

```
alias i='sudo apt install'
alias cpu10='ps -L aux | sort -nr -k 3 | head -10'
alias mem10='ps -L aux | sort -nr -k 4 | head -10'
alias lsmount='mount | column -t'
```

There are, of course, additional time-saving tricks that we could talk about here, but then again, Bash is so complex that we can write an entire book about it (and many people have). As we go along in this chapter, I'll give you even more tips. For now, here's a final trick, which changes your working directory back to the previous directory you were in:

```
cd -
```

That simple command was mentioned in *Chapter 4, Navigating and Essential Commands*, but it's worth a second mention—you're welcome! Next, let's take a look at shell variables, which allow us to store information for easy access in other commands.

Understanding variables

Bash is more than just a shell. You could argue that it is very similar to a complete programming language, and you wouldn't be wrong. Bash has a built-in scripting engine (we will get into scripting later) and there are many debates as to what distinguishes a scripting language from a programming language, and that line becomes more and more blurred as new languages come out.

As with any scripting language, Bash supports variables. The concept of variables is very simple in Bash, but I figured I'd give it its own (relatively short) section to make sure you understand the basics. You can set a variable with a command such as the following:

```
myvar='Hello world!'
```

When Bash encounters an equal sign after a string, it assumes you're creating a variable. Here, we're creating a variable named `myvar` and setting it equal to `Hello world!` Whenever we refer to a variable, though, we need to specifically clarify to Bash that we're requesting a variable, and we do that by prefixing it with a dollar symbol (`$`). Consider this command:

```
echo $myvar
```

If you've set the variable as I have, executing that command will print `Hello world!` to `stdout`. The `echo` command is very useful for printing the contents of variables. The key thing to remember here is that when you *set* a variable, you don't include the `$` symbol, but you do when you retrieve it. Also, keep in mind that there's no space on either side of the equals sign.

You will see variations of variable name formats as you work with various Linux servers. For example, you may see variable names in all caps, camel case (`MyVar`), as well as other variations. These variations are all valid, and depending on the background of the individual creating them (developers, administrators, and so on), you may see different forms of variable naming. It's also extremely common that built-in variables created as part of the distribution will be named in all upper case.

Variables work in other aspects of the shell as well, not just with `echo`. Consider this:

```
mydir="/etc"  
ls $mydir
```

Here, we're storing a directory name in a variable and using the `ls` command against it to list its contents. This may seem relatively useless, but when you're scripting, this will save you time. Anytime you need to refer to something more than once, it should be in a variable. That way, in a script, you can change the contents of that variable just one time and everywhere in the script will reference it.

There are also variables that are automatically present in your shell that you did not explicitly set yourself. Enter this command for fun:

```
env
```

Wow! You should see a lot of variables, especially if you enter it in a desktop version of Ubuntu.

These variables are set by the system, but can still be accessed via `echo` as you would any other. Some notable ones include `$SHELL` (stores the name of the binary that currently handles your shell), `$USER` (stores your current username), and `$HOST` (stores the hostname for your device). Any of these variables can be accessed at any time, and may even prove beneficial in scripts.

We've already gone over **standard output (stdout)**, **standard error (stderr)**, and **standard input (stdin)** in the previous chapter. We'll use standard input again here, when we capture input to store it as a variable. Try this command for example:

```
read age
```

When you run this command, you'll just be brought to a blank line, with no indication as to what you should be doing. Go ahead and enter your age, and then press *Enter*. Next, run this:

```
echo $age
```

In a script, you would want to inform the user what they should be entering, so you would probably use something similar to these commands:

```
echo "Please enter your age"  
read age  
echo "Your age is $age"
```

We've discussed standard input in the previous chapter, and we can see it in action again here as we capture input from the user and store it in a variable.

Automation is a subject we'll explore multiple times throughout the remainder of the book, which will include more advanced subjects such as configuration management. Writing scripts is the simplest form of automation, which gives you the ability to type commands in a text file and have them all execute. That's what we'll explore next.

Writing simple scripts

This is the section where everything we've talked about so far starts to come together. Writing scripts can be very fun and rewarding, as they allow you to automate large jobs or just simplify something that you find yourself doing over and over. The most important point about scripting is this: if it's something you'll be doing more than once, you really should be making it into a script. This is a great habit to get into.

A script is a very simple concept; it's just a text file that contains commands for your shell to execute one by one. A script written to be executed by Bash is known as a Bash script, and that's what we'll work on creating in this section.

At this point, I'm assuming that you've practiced a bit with a text editor in Linux. It doesn't matter if you use Vim or Nano. Since we've edited text files before (we went over that in *Chapter 5, Managing Files and Directories*), I'm under the assumption that you already know how to create and edit files. We'll be using a text editor to create a simple script as an example, using the following command:

```
nano ~/myscript.sh
```

If you weren't already aware, a tilde (~) is just a shortcut for a user's home directory. Therefore, on my system, the previous command would be the same as if I had typed:

```
nano /home/jay/myscript.sh
```

Inside the file, type the following:

```
#!/bin/bash
echo "My name is $USER"
echo "My home directory is $HOME"
```

Save the file and exit the editor. In order to run this file as a script, we need to mark it as executable:

```
chmod +x ~/myscript.sh
```

To execute it, we simply call the path to the file and the filename:

```
~/myscript.sh
```

The output should look similar to the following:

```
My name is jay
My home directory is /home/jay
```

The first line, `#!/bin/bash`, might seem strange if you haven't seen it before. Normally, lines starting with a hash symbol (`#`) are ignored by the interpreter. The one on the first line is an exception to this. The `#!/bin/bash` entry we see on the first line is known as a **hash bang**, or **shebang**. Basically, it just tells the kernel which interpreter to use in order to run the commands inside the script. There are other interpreters we could be using, such as `#!/usr/bin/python` if we were writing a script in the Python language. Since we're writing a Bash script, we used `#!/bin/bash`.

The lines that followed were simple print statements. Each one used a system variable, so you didn't have to declare any of those variables as they already existed. Here, we printed the current user's username and home directory.

The concept of scripting becomes more valuable when you start to think of things you do on a regular basis that you can instead automate. To be an effective Linux administrator, it's important to adopt the automation mindset. Again, if you are going to do a job more than once, script it. Here's another example script to help drive this concept home. This time, the script will actually be somewhat useful:

```
#!/bin/bash
sudo apt install -y apache2
sudo apt install -y libapache2-mod-php8.1
sudo a2enmod php8.1
sudo systemctl restart apache2
```

What we've done here is theoretically scripted the setup of a web server. We could extend this script further by having it copy site content to `/var/www/html`, enable a configuration file, and so on. But from the preceding script, you can probably see how scripting can be useful in condensing the amount of work you do. This script could be an advanced web server install script that you could simply copy to a new server and then run.

Notice that the example uses the `-y` option with `apt`. If you weren't already aware, that automatically answers *yes* to prompts that may come up as part of the process. Scripts are typically not interactive, meaning there may not be an administrator sitting in front of it to answer prompts when they appear. Also, using the `a2enmod` command to enable `php8.1` was not really necessary, as it would've been enabled automatically as part of installing the `libapache2-mod-php8.1` package. But I think you get the idea; we want to be explicit in scripts and type instructions for the exact state we want things to be in.

Now, let's get a bit more advanced with scripting. The previous script only installed some packages, something we probably could've done just as easily by copying and pasting the commands into the shell. Let's take this script a bit further. Let's write a conditional statement. Here's a modified version of the previous script:

```
#!/bin/bash
# Install Apache if it's not already present
if [ ! -f /usr/sbin/apache2 ]; then
    sudo apt install -y apache2
    sudo apt install -y libapache2-mod-php8.1
    sudo a2enmod php8.1
    sudo systemctl restart apache2
fi
```

Now it's getting a bit more interesting. The first line after the hash bang is a comment, letting us know what the script does:

```
# Install Apache if it's not already present
```

Comments are ignored by the interpreter, but are useful in letting us know what a block of code is doing.

Next, we start an `if` statement:

```
if [ ! -f /usr/sbin/apache2 ]; then
```

Bash, like any scripting language, supports branching and the `if` statement is one way of doing that. Here, it's checking for the existence of the `apache2` binary. The `-f` option here specifies that we're looking for a file. We can change this to `-d` to check for the existence of a directory instead. The exclamation mark is an inverse. It basically means we're checking if something is *not* present. If we wanted to check if something *is* present, we would omit the exclamation mark. Basically, we're setting up the script to do nothing if Apache is already installed. In this case, inside the brackets we are just executing a shell command, and then the result is checked. The commands sandwiched inside the `if` statement are simply installing packages.

Finally, we close out our `if` statement with the word *if* backward (`fi`). If you forget to do this, the script will fail.

With regard to the concept of `if` statements, we can compare values as well. Consider the following example:

```
#!/bin/bash
```

```
myvar=1
if [ $myvar -eq 1]; then
    echo "The variable equals 1"
fi
```

With this script, we're merely checking the contents of a variable, and taking action if it equals a certain number. Notice we didn't use quotation marks when creating the variable, since we just set a number (integer) here. We would've only used quotation marks if we wanted to set the variable value to a string. We can also take action if the `if` statement doesn't match:

```
#!/bin/bash
myvar=10
if [ $myvar -eq 1]; then
    echo "The variable equals 1"
else
    echo "The variable doesn't equal 1"
fi
```

This was a silly example, I know, but it works as far as illustrating how to create an `if/else` logic block in Bash. The `if` statement checks to see whether the variable was equal to 1. It isn't, so the `else` block executes instead.

The `-eq` portion of the command is similar to `==` in most programming languages. It's checking to see whether the value is equal to something. Alternatively, we can use `-ne` (not equal), `-gt` (greater than), `-ge` (greater than or equal to), `-lt` (less than), and so on.

At this point, I recommend you take a break from reading to further practice scripting (practice is key to committing concepts to memory). Try the following challenges:

- Ask the user to enter input, such as their age, and save it to a variable. If the user enters a number less than 30, tell them they're young. If the number is equal to or greater than 30, echo a statement telling them that they're old.
- Write a script that copies a file from one place to another. Make the script check to see whether that file exists first, and have an `else` statement printing an error if the file doesn't exist.
- Think about any topic we've already worked on during this book, and attempt to automate it.

Now, let's take a look at another concept, which is looping. The basic idea behind looping is simply doing something repeatedly until a condition has been met. Consider the following example script:

```
#!/bin/bash
myvar=1
while [ $myvar -le 15 ]
do
    echo $myvar
    ((myvar++))
done
```

Let's go through the script line by line to understand what it's doing.

```
myvar=1
```

With this new script, we're creating a control variable, called `myvar`, and setting it equal to 1.

```
while [ $myvar -le 15 ]
```

Next, we set up a `while` loop. A `while` loop will continue until a condition is met. Here, we're telling it to execute the statements in the block over and over until `$myvar` becomes equal to 15. In fact, a `while` loop can continue forever if you enter something incorrectly, which is known as an **infinite loop**. An infinite loop is dangerous, and can cause your server to stop responding. If you used `-ge 0` instead, you would've created exactly that.

```
do
```

With `do`, we're telling the `for` loop to prepare itself to start doing something.

```
echo $myvar
```

Here, we're printing the current content of the `$myvar` variable—nothing surprising here.

```
((myvar++))
```

With this statement, we're using what's known as an `incrementor` to increase the value of our variable by 1. The double parenthesis tells the shell that we're doing an arithmetic operation, so the interpreter doesn't think that we're working with strings.

```
done
```

When we're done writing a `while` loop, we must close the block with `done`. If you've typed the script properly, it should count from 1 to 15.

Another type of loop is a `for` loop. A `for` loop executes a statement for every item in a set. For example, you can have the `for` loop execute a command against every file in a directory. Consider this example:

```
#!/bin/bash
turtles='Donatello Leonardo Michelangelo Raphael'
for t in $turtles
do
    echo $t
done
```

Let's take a deeper look into what we've done here:

```
turtles='Donatello Leonardo Michelangelo Raphael'
```

Here, we're creating a list and populating it with names. Each name is one item in the list. We're calling this list `turtles`. We can see the contents of this list with `echo` as we would with any other variable:

```
echo $turtles
```

Next, let's look at how we set up the for loop:

```
for t in $turtles
```

Now, we're telling the interpreter to prepare to do something for every item in the list. The `t` here is arbitrary, we could've used any letter here or even a longer string. We're just setting up a temporary variable we want to use in order to hold the current item the script is working on.

```
do
```

With `do`, we're telling the for loop to prepare itself to start doing something.

```
echo $t
```

Now, we're printing the current value of `$t` to `stdout`.

```
done
```

Just as we did with the `while` loop, we type `done` to let the interpreter know this is the end of the for loop. Effectively, we just created a for loop to print each item in a list independently:

```
Donatello
Leonardo
Michelangelo
Raphael
```

We included four turtle names in our list, and we were able to iterate through them and print them out, one by one.

As much as I love turtles (especially the teenage mutant ninja variety), that script isn't very practical or useful to us for server administration. Next, we're going to write a script that can actually be quite useful.

Putting it all together – Writing an `rsync` backup script

Let's close this chapter with a Bash script that will not only prove to be very useful but will also help you enhance your skills. The `rsync` utility is one of my favorites; it's very useful for copying data from one place to another, as well as being helpful for setting up a backup job. Let's use the following example `rsync` command to practice automation:

```
rsync -avb --delete --backup-dir=/backup/incremental/08-17-2022 /src /target
```

This example `rsync` command uses the `-a` (archive) option, which retains the metadata of the file(s) it copies to the target, such as the timestamp and owner. The `-v` option gives us verbose output, so we can see exactly what `rsync` is doing. The `-b` option enables backup mode, which means that if a file on the target will be overwritten by a file from the source, the previous version of that file will be renamed so it won't be overwritten. Combining these three options, we simplify it to `-avb` rather than typing `-a -v -b`. The `--delete` option tells `rsync` to delete any files in the target that aren't present in the source (since we used `-b`, any file that is deleted will be retained).

The `--backup-dir` option tells `rsync` that any time a file would have been renamed in this way (or deleted), to instead just copy it to another directory. In this case, we send any files that would have been overwritten to the `/backup/incremental/08-16-2022` directory.

Let's script this `rsync` job. One problem we can fix in our script right away is the date that is present inside the directory we're using for the `--backup-dir`. The date changes every day, so we shouldn't be hardcoding this. Therefore, let's start our script by addressing this:

```
#!/bin/bash
curdate=$(date +%m-%d-%Y)
```

We're creating a variable called `curdate`. We're setting it equal to the output of the `$(date +%m-%d-%Y)` command. You can execute `date +%m-%d-%Y` in your terminal window to see exactly what that does. In this case, putting a command (such as `date`) in parentheses and a dollar symbol means that we're executing the command in a **sub-shell**. The command will run, and we're going to capture the result of that command and store it in the `curdate` variable.

Next, let's make sure `rsync` is actually installed, and install it if it's not:

```
if [ ! -f /usr/bin/rsync ]; then
    sudo apt install -y rsync
fi
```

Here, we're simply checking to see whether `rsync` is *not* installed. If it's not, we'll install it via `apt`. This is similar to how we checked for the existence of `apache2` earlier in this chapter.

Now, we add the final line:

```
rsync -avb --delete --backup-dir=/backup/incremental/$curdate /src /target
```

You can definitely see the magic of variables in Bash now, if you haven't already. We're including `$curdate` in our command, which is set to whatever the current date actually is. When we put it all together, our script looks like this:

```
#!/bin/bash
curdate=$(date +%m-%d-%Y)
if [ ! -f /usr/bin/rsync ]; then
    sudo apt install -y rsync
fi
rsync -avb --delete --backup-dir=/backup/incremental/$curdate /src /target
```

This script, when run, will run an `rsync` job that will copy the contents from `/src` to `/target`. (Be sure to change these directories to match the source directory you want to back up and the target where you want to copy it to.) The beauty of this is that `/target` can be an external hard drive or network share. So, in a nutshell, you can automate a nightly backup. This backup, since we used the `-b` option along with `--backup-dir`, will allow you to retrieve previous versions of a file from the `/backup/incremental` directory. Feel free to get creative here as far as where to place previous file versions and where to send the backup.

Of course, don't forget to mark the script as executable, assuming it was saved with a name like `backup.sh`:

```
chmod +x backup.sh
```

At this point, you can put this script in a cron job to automate its run. To do so, it's best to put the script in a central location where it can be found, such as in `/usr/local/bin`:

```
sudo mv backup.sh /usr/local/bin
```

You can consider creating a cron job for this script to be run periodically. We'll cover that in *Chapter 7, Controlling and Monitoring Processes*, which is the very next chapter. With a cron job, you can set up various tasks to run at different times in order to make your server essentially do your work for you.

Summary

In this chapter, we dived into a number of more advanced concepts relating to shell commands, such as redirection, Bash history, command aliases, some command-line tricks, and more. Working with the shell is definitely something you'll continue to improve upon, so don't be worried if you have any trouble committing all of this knowledge to memory. After over 20 years of working with Linux, I'm still learning new things myself. The main takeaway in this chapter is to serve as a starting point to broaden your command-line techniques and also serve as the basis for future exploration into the subject.

In the next chapter, we'll take a look at managing processes, which will include job management, taming misbehaving processes, and more. See you there!

Relevant videos

- Linux Crash Course – Bash History (LearnLinuxTV): <https://linux.video/le-bash-history>
- Linux Command-Line Tips & Tricks (LearnLinuxTV): <https://linux.video/cli-tips>

Further reading

- Comparison operators for Bash: <https://learnlinux.link/c-ops>
- Commandlinefu: <https://learnlinux.link/c-fu>
- Bash reference manual: <https://learnlinux.link/bash-man>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



7

Controlling and Managing Processes

On a typical Linux server, there can be over a hundred processes running at any given time. The purposes of these processes range from system services, such as the **Network Time Protocol (NTP)** service, to processes that serve information to others, such as the Apache web server. As an administrator of Ubuntu servers, you will need to be able to manage these processes, as well as managing the resources available to them. In this chapter, we'll take a look at process management, including the `ps` command, managing job control commands, and more.

As we work through these concepts, we will cover the following topics:

- Managing jobs
- Understanding the `ps` command
- Changing the priority of processes
- Dealing with misbehaving processes
- Managing system processes
- Scheduling tasks with `cron`

To begin our exploration of managing processes, let's take a look first at managing jobs. Not only will this help us understand the concepts better, but it will also provide us with a better understanding of backgrounding and foregrounding.

Managing jobs

Up until now, everything we have been doing on the shell has been right in front of us, from execution to completion. We've installed applications, run programs, and walked through various commands. Each time, we've had control of our shell taken from us, and we've only been able to start a new task when the previous one had finished. For example, if we were to install the `vim-nox` package with the `apt install` command, we would watch helplessly while `apt` takes care of fetching the package and installing it for us.

While this is going on, our cursor goes away and our shell completes the task for us without allowing us to queue up another command. We can always open a new shell to the server and multitask by having two windows open at once, each doing different tasks. But that's likely not going to be the most efficient method of multitasking when working with the command line.

Instead, we can actually background a process without waiting for it to complete while working on something else. Then, we can bring that process back to the front to return to working on it or to check whether or not it finished successfully. Think of this as a similar concept to a windowing desktop environment, or user interfaces on the Windows or macOS operating systems. We can work on an application, minimize it to get it out of the way, and then maximize it to continue working with it. Essentially, that's the same concept as backgrounding a process in a Linux shell.

So how exactly do you background and foreground a process? This concept can be somewhat difficult to explain. In my opinion, the easiest way to learn a new concept is to try it out, and the easiest example I can think of is by (yet again) using a text editor. I promise that this time, using a text editor as an example won't be boring. In fact, this example is extremely useful and may just become a part of your daily workflow. To do this exercise, you can use any command-line text editor you prefer, such as Vim or Nano. On Ubuntu Server, nano is usually installed by default, so you already have it if you want to go with that. If you prefer to use Vim, feel free to install the `vim-nox` package if you haven't already installed it:

```
sudo apt install vim-nox
```

You can actually install `vim` rather than `vim-nox`, but I always default to `vim-nox` since it features built-in support for scripting languages.

Again, feel free to use whichever text editor you feel comfortable with. In the following examples, I'll be using nano, but if you use vim, just replace nano with vim every time you see it.

Anyway, to see backgrounding in action, open up your text editor. Feel free to open a file or just start a blank session. (If in doubt, type nano and press *Enter*.) With the text editor open, we can background it at any time by pressing `Ctrl + z` on our keyboard.

If you are using vim instead of nano, you can only background vim when you are *not* in *insert mode*, since it captures `Ctrl + z` rather than passing it to the shell.

Did you see what happened? You were immediately taken away from your editor and returned to the shell so you can now get back to executing commands. You should have seen some output similar to the following:

```
[1]+ Stopped nano
```

Here, we see the job number of our process, its status, and then the name of the process. Even though the process of your text editor shows a status of Stopped, it's still running. You can confirm this with the following command:

```
ps au | grep nano
```

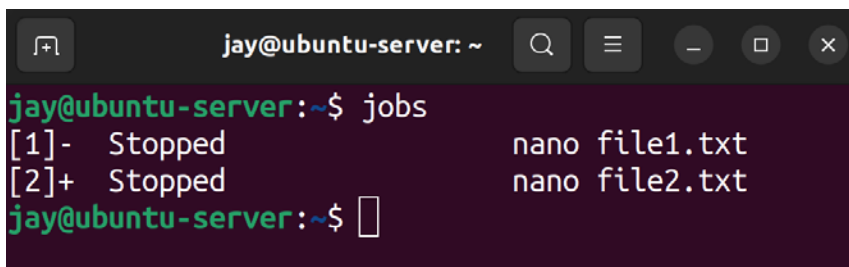
In my case, I see the nano process running with a PID of 43231:

```
jay      43231  0.0  0.1  5468  3632 pts/0    T   11:27   0:00 nano
```

At this point, I can execute additional commands, navigate around my filesystem, and get additional work done. When I want to bring my text editor back, I can use the `fg` command to foreground the process, which will resume it. If I have multiple background processes, the `fg` command will bring back the one I was working on most recently.

I gave you an example of the `ps` command to show that the process was still running in the background, but there's actually a dedicated command for that purpose, and that is the `jobs` command.

If you execute the `jobs` command, you'll see in the output a list of all the processes running in the background:



```

jay@ubuntu-server: ~
┌───┴───┐
jay@ubuntu-server:~$ jobs
[1]-  Stopped          nano file1.txt
[2]+  Stopped          nano file2.txt
jay@ubuntu-server:~$ █

```

Figure 7.1: Running the `jobs` command after backgrounding two nano processes

The output shows that I have two nano sessions in use, one modifying `file1.txt`, and the other modifying `file2.txt`. If I were to execute the `fg` command, that would bring up the nano session that's editing `file2.txt`, since that was the last one I was working in. That may or may not be the one I want to return to editing, though. Since I have the job ID on the left, I can bring up a specific background process by using its ID with the `fg` command:

```
fg 1
```

Knowing how to background a process can add quite a bit to your workflow. For example, let's say, hypothetically, that I'm editing a config file for a server application, such as Apache. While I'm editing this config file, I need to consult the documentation (man page) for Apache because I forgot the syntax for something. I could open a new shell and an SSH session to my server and view the documentation in another window. This could get very messy if I open up too many shells. It would be much simpler to background the current nano session, read the documentation, and then foreground the process with the `fg` command to return to working on it, all from one SSH session!

To background a process, you don't have to use `Ctrl + z`; you can actually background a process right when you execute it by entering a command with the ampersand symbol (`&`) typed at the end. To show you how this works, I'll use `htop` as an example. Admittedly, this may not necessarily be the most practical example, but it does work to show you how to start a process and have it backgrounded right away.

We might not have `htop` installed yet, but for now, feel free to install this package (if it isn't already) and then run it with the ampersand symbol:

```
sudo apt install htop
htop &
```

The first command, as you already know, installs the `htop` package on our server. With the second command, I'm opening `htop` but backgrounding it immediately. What I'll see when it's backgrounded is its job ID and process ID (more on this in the next section). Now, at any time, I can bring `htop` to the foreground with `fg`. Since I just backgrounded it, `fg` will bring `htop` back since it considers it the most recent. As you know, if it wasn't the most recent, I could reference its job ID with the `fg` command to bring it back even if it wasn't my most recently used job. Go ahead and practice using the ampersand symbol with a command and then bringing it back to the foreground. In the case of `htop`, it can be useful to start it, background it, and then bring it back anytime you need to check the performance of your server.

Keep in mind, though, that when you exit your shell, all your backgrounded processes will close. If you have unsaved work in your text editors, you'll lose what you were working on. For this reason, if you utilize background processes, you may want to check to see if you have any pending jobs still running by executing the `jobs` command before logging out.

In addition, you'll probably notice that some applications background cleanly, while others don't. In the case of using a text editor and `htop`, those applications stay paused in the background, allowing us to perform other tasks and then return to those commands later. However, some applications may still spit out diagnostic text regularly in your main window, whether they're backgrounded or not. To get even more control over your Bash sessions, you can learn how to use a multiplexer, such as `tmux` or `screen`, to allow these processes to run in their own session such that they don't interrupt your work. Going over the use of a program such as `tmux` is beyond the scope of this book, but it is a useful utility to learn if you're interested.

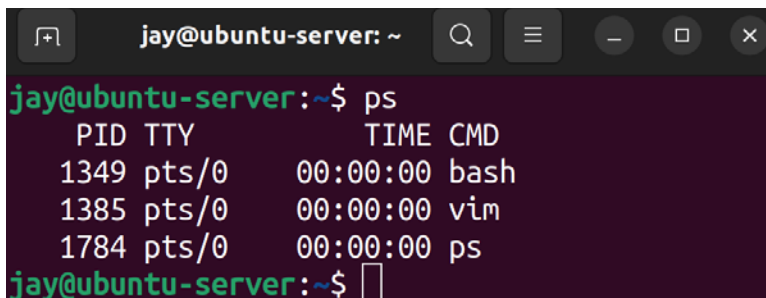
Being able to background and foreground a process allows us to manage tasks on the command line more effectively, and is definitely useful. Now, we can expand this and look at viewing other processes on the server, including those that we didn't manually start such as a text editor. In the next section, we'll take a look at the `ps` command, which can help us understand what is actually running on our server.

Understanding the `ps` command

While managing our server, we'll need to understand what processes are running and how to manage them. Later in this chapter, we'll work through starting, stopping, and monitoring processes. But before we get to those concepts, we first need to be able to determine what is actually running on our server. The `ps` command allows us to do this.

Viewing running processes with `ps`

When executed by itself, the `ps` command will show a list of processes run by the user who called the command:

A terminal window titled 'jay@ubuntu-server: ~' with standard window controls. The prompt is 'jay@ubuntu-server:~\$'. The command 'ps' has been entered and executed. The output is a table with four columns: PID, TTY, TIME, and CMD. The first three rows represent running processes: PID 1349 (bash), PID 1385 (vim), and PID 1784 (ps). The prompt 'jay@ubuntu-server:~\$' is visible again at the bottom with a cursor.

```
jay@ubuntu-server:~$ ps
  PID TTY          TIME CMD
 1349 pts/0    00:00:00 bash
 1385 pts/0    00:00:00 vim
 1784 pts/0    00:00:00 ps
jay@ubuntu-server:~$
```

Figure 7.2: The output of the `ps` command, when run as a normal user and with no options

In Figure 7.2, you can see that when I ran the `ps` command as my own user with no options, it showed me a list of processes that I am running as myself. In this case, I have a `vim` session open (running in the background), and in the last line, we also see `ps` itself, which is also included in the output.

On the left side of the output, you'll see a number for each of the running processes. This is known as the **Process ID (PID)**, which we mentioned in the *Managing jobs* section. Before we continue on, the PID is something that you really should be familiar with, so we may as well cover it right now.

Each process running on your server is assigned a PID, which differentiates it from other processes on your system. You may understand a process as `vim`, or `top`, or some other name. However, our server knows processes by their ID. When you open a program or start a process, it's given a PID by the kernel. As you work on managing your server, you'll find that the PID is useful to know, especially for the commands we'll be covering in this very chapter. If you want to kill a misbehaving process, for example, a typical workflow would be for you to find the PID of that process and then reference that PID when you go to kill the process (which I'll show you how to do in a later section). PIDs are actually more complex than just a number assigned to running processes, but for the purposes of this chapter, that's the main purpose we'll need to remember.

You can also use the `pidof` command to find the PID of a process if you know the name of it. For example, I showed you a screenshot of a `vim` process running with a PID of 1385. You can also do so by running the following command:

```
pidof vim
```

The output will give you the PID(s) of the process without you having to use the `ps` command.

Configuring arguments to `ps`

Continuing with the `ps` command, there are several useful arguments you can give in order to change the way in which it produces an output. If you use the `a` option, you'll see more information than you normally would:

```
ps a
```

This will produce an output something like the following:

```

jay@ubuntu-server: ~$ ps a
  PID TTY          STAT       TIME COMMAND
   774 tty1      Ss+        0:00 /sbin/agetty -o -
  1349 pts/0    Ss         0:00 -bash
  1789 pts/0    T          0:00 vim testfile.txt
  1790 pts/0    R+        0:00 ps a
jay@ubuntu-server: ~$

```

Figure 7.3: The output of the `ps a` command

With `ps a`, we’re seeing the same output as before, but with additional information, as well as column headings at the top. We now see a heading for PID, TTY, STAT, TIME, and COMMAND. From this new output, you can see that the `vim` processes I have running are editing a file named `testfile.txt`. This is great to know, because if I had more than one `vim` session open and one of them was misbehaving, I would probably want to know which one I specifically needed to stop.

We already saw the PID and COMMAND fields, although we didn’t see a formal heading at the top. The PID column we’ve already covered, so I won’t go into any additional detail about that. The COMMAND field tells us the actual command being run, which is very useful if we either want to ensure we’re managing the correct process or to see what a particular user is running (I’ll demonstrate how to display processes for other users soon).

The STAT, field is new; we didn’t see it when we ran `ps` by itself. The STAT field gives us the status code of the process, which refers to which state the process is currently in. The state can be uninterruptible sleep (D), defunct (Z), stopped (T), interruptible sleep (S), and in the run queue (R). There is also paging (w), but that is not used anymore, so there’s no need to cover it. Uninterruptible sleep is a state in which a process is generally waiting on input and cannot handle additional signals (we’ll briefly talk about signals later on in this chapter). A defunct process (also referred to as a zombie process) has, for all intents and purposes, finished its job but is waiting on the parent to perform cleanup. Defunct processes aren’t actually running, but remain in the process list and should normally close on their own. If such a process remains in the list indefinitely and doesn’t close, it can be a candidate for the `kill` command, which we will discuss later. A stopped process is generally a process that has been sent to the background, which will be discussed in the next section. Interruptible sleep means that the program is idle: it’s waiting for input in order to awaken.

The TTY column tells us which TTY the process is attached to. A TTY refers to a **teletypewriter**, which is a term used from a much different time period. In the past, during the time of big mainframes, users would use such computers using “terminals” – a device consisting of a monitor and keyboard, connected (via the wire) to the mainframe. Such devices could only display the output received from the mainframe, and receive data typed on the keyboard. Teletypewriter was the term used to refer to such devices. Obviously, we don’t use machines like these nowadays, but the concept is similar from a virtual standpoint.

On our server, we're using our keyboard to send input to a device that then displays output to another device. In our case, the input device is our keyboard and the output device is our screen, which is either connected directly to our server or is located on our computer, which is connected to our server over a service such as SSH. On a Linux system, most processes run on a TTY, which is (for all intents and purposes) a terminal that grabs input and manages the output, similar to a teletypewriter in a virtual sense. A terminal is our method of interacting with our server.

In *Figure 7.3*, we have a process running on a TTY of `tty1`, and the other processes are running on `pts/0`. The TTY we see is the actual terminal device, and `pts` references a virtual (pseudo) terminal device. Our server is actually able to run several `tty` sessions, typically one to seven. Each of these can be running its own programs and processes. To understand this better, try pressing `Ctrl + Alt +` any function key, from `F1` through `F7` (if you have a physical keyboard plugged into a physical server). Each time, you should see your screen cleared and then moved to another terminal. Each of these terminals is independent of one another. Each of your function keys represents a specific TTY, so by pressing `Ctrl + Alt + F6`, you're switching your display to TTY 6.

Essentially, you're switching from TTY 1 through to TTY 7, with each being able to contain its own running processes. If you run `ps` again, you'll see any processes you start on those TTYs show up in the output as a `tty` session, such as `tty2` or `tty4`. Processes that you start in a terminal emulator will be given a designation of `pts`, because they're not running in an actual TTY, but rather a pseudo-TTY.

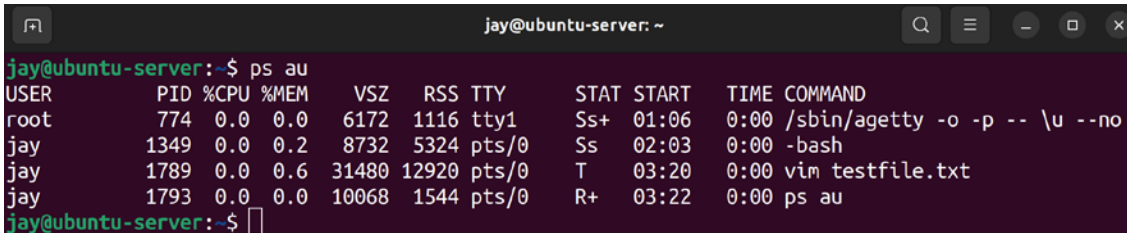
This was a long discussion for something that ends up being simple (TTY or pseudo-TTY), but with this knowledge, you should be able to differentiate between a process running on the actual server or through a shell.

Continuing, let's take a look at the `TIME` field of our `ps` command output. This field represents the total amount of time the CPU has been utilized for that particular process. However, the time is `0:00` for each of the processes in the screenshot I've provided. This may be confusing at first. In my case, the `vim` processes in particular have been running for about 15 minutes or so since I took the screenshot, and they still show `0:00` utilization time even now. Actually, this isn't the amount of time the process has been running, but rather the amount of time the process has been actively engaging with the CPU. In the case of `vim`, each of these processes is just a buffer with a file open. For the sake of comparison, the Linux machine I'm writing this chapter on has a process ID of 759 with a time of `92:51`. PID 759 belongs to my X server, which provides the foundation for my **graphical user interface (GUI)** and windowing capabilities. However, this laptop currently has an uptime of 6 days and 22 hours as I type this, which is roughly equivalent to 166 hours, which is not the same amount of time that PID 759 is reporting in its `TIME` entry. Therefore, we can deduce that even though my laptop has been running for 6 days straight, the X server has only utilized 92 hours and 51 minutes of actual CPU time. In summary, the `TIME` column refers to the amount of time a process needs the CPU in order to calculate something and is not necessarily equal to how long something has been running, or for how long a graphical process is showing on your screen.

Let's continue on with the `ps` command and look at some additional options. First, let's see what we get when we add the `u` option to our previous example, which gives us the following example command:

```
ps au
```

This will produce an output that will look similar to the following:



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ ps au
USER      PID %CPU %MEM    VSZ   RSS TTY      STAT START   TIME COMMAND
root         774  0.0  0.0  6172  1116 tty1    Ss+  01:06   0:00 /sbin/agetty -o -p -- \u --no
jay       1349  0.0  0.2   8732   5324 pts/0    Ss   02:03   0:00 -bash
jay       1789  0.0  0.6  31480 12920 pts/0    T   03:20   0:00 vim testfile.txt
jay       1793  0.0  0.0  10068   1544 pts/0    R+  03:22   0:00 ps au
jay@ubuntu-server:~$

```

Figure 7.4: The output of the `ps au` command

When you run it, you should notice the difference from the `ps a` command right away. With this variation, you'll see processes listed that are being run by your user ID, as well as other users. When I run it, I see processes listed in the output for my user (jay), as well as one for root. The `u` option will be a common option you're likely to use, since most of the time while managing servers, you're probably more interested in keeping an eye on what kinds of shenanigans your users are getting themselves into. But perhaps the most common use of the `ps` command is the following variation:

```
ps aux
```

With the `x` option added, we're no longer limiting our output to processes within a TTY (either native or pseudo). The result is that we'll see a lot more processes, including system-level processes that are not tied to a process we started ourselves. Go ahead and try it. In practice, though, the `ps aux` command is most commonly used with `grep` to look for a particular process or string. For example, let's say you want to see a list of all `nginx` worker processes. To do that, you may execute a command such as the following:

```
ps aux | grep nginx
```

Here, we're executing the `ps aux` command as before, but we're piping the output into `grep`, where we're looking only for lines of output that include the string `nginx`. In practice, this is the way I often use `ps`, as well as the way I've noticed many other administrators using it. With `ps aux`, we are able to see a lot more output, and then we can narrow that down with search criteria by piping into `grep`. However, if all we wanted to do was to show processes that have a particular string, we could also do the following:

```
ps u -C nginx
```

This would produce output containing a list of processes matching `nginx`, and related details. Another useful variation of the `ps` command is to sort the output by sorting the processes using the most CPU first:

```
ps aux --sort=-pcpu
```


Unfortunately, that command shows a lot of output, and we would have to scroll back to the top in order to see the top processes. Depending on your terminal, you may not have the ability to scroll back very far (or at all), so the following command will narrow it down further:

```
ps aux --sort=-pcpu | head -n 5
```

Now that is useful! With that example, I'm using the `ps aux` command with the `--sort` option, sorting by the percentage of CPU utilization (`-pcpu`). Then I'm piping the output into the `head` command, where I'm instructing it to show me only five lines (`-n 5`). Essentially, this is giving me a list of the top five processes that have used the most CPU since boot time. In fact, I can do the same, but with the most-used memory instead:

```
ps aux --sort=-pmem | head -n 5
```

If you want to determine which processes are misbehaving and using a non-ordinary amount of memory or CPU, those commands will help you narrow it down. The `ps` command is a very useful command for your admin toolbox. Feel free to experiment with it beyond the examples I've provided; you can consult the man pages for the `ps` command to learn even more tricks. In fact, the second section of the man page for `ps` (under *examples*) gives you even more neat examples to try out.

Now that we know how to inspect running processes, in the next section, we'll take a look at how to change the priority of the processes to ensure those that are more important are given extra attention by the CPU.

Changing the priority of processes

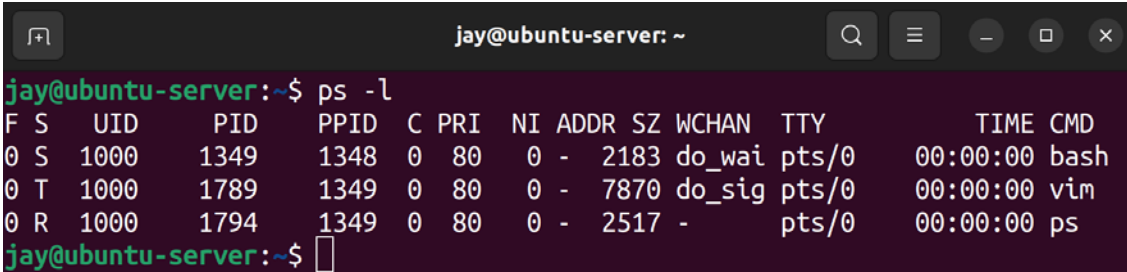
Processes on a Linux system can be run with an altered priority, giving some processes more priority and others less. This gives you, the administrator, full reign when it comes to ensuring that the most important processes on the system are running with an adequate level of prioritization. There are dedicated commands for this purpose: `nice` and `renice`. These commands allow you to launch a process with a specific priority, or change the priority of a process that's already running.

Nowadays, manually editing the priority of a process is something administrators will find themselves doing less often than they used to. A processor with 32 cores (or many more) is not all that uncommon, and neither is hundreds of gigabytes of RAM. Servers nowadays are certainly more powerful than they used to be, and are nowhere near as resource-starved as machines of old. Many servers (such as virtual machines) and containers are dedicated to a single task, so process tuning may not be of extreme value anymore. However, data processing firms and companies utilizing deep learning functions may find themselves needing to fine-tune some things.

Regardless of whether or not prioritizing processes is something that will be immediately useful to you, it's a good idea to at least understand the concept just in case you do find yourself needing to increase or decrease the priority of a process some day. Let's revisit the `ps` command, this time with the `-l` argument:

```
ps -l
```


The output of this command will appear as follows:



```

jay@ubuntu-server:~$ ps -l
F S  UID      PID     PPID  C  PRI  NI ADDR  SZ  WCHAN  TTY          TIME CMD
0 S  1000    1349    1348  0   80   0  - 2183 do_wai pts/0      00:00:00 bash
0 T  1000    1789    1349  0   80   0  - 7870 do_sig pts/0      00:00:00 vim
0 R  1000    1794    1349  0   80   0  - 2517 -      pts/0      00:00:00 ps
jay@ubuntu-server:~$

```

Figure 7.5: The output of the `ps -l` command

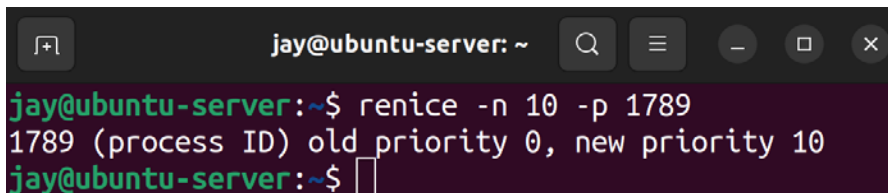
With the output of the `ps -l` command, notice the `PRI` and `NI` columns. `PRI` refers to the priority, and `NI` pertains to the “niceness” value, which we’ll discuss in more detail later in this section. In this example, each process that I’m running has a `PRI` of `80`, and an `NI` of `0`. I didn’t change or alter any of these; these are the values that I get when I start processes with no special tweaks. A `PRI` value of `80` is the starting value for that value on all processes, and will change as we increase or decrease the niceness value.

As I mentioned, we have dedicated commands that allow us to alter priorities, `nice` and `renice`. To determine which to use, it all comes down to whether or not the process is already running. With regard to the processes listed in *Figure 7.5*, we would want to use `renice` to change the priority for those, since they’re all already running. If we wanted to launch a process with a specific priority right from the beginning, we would use `nice` instead.

For example, let’s change the process of the `vim` session I have running. Sure, this is a somewhat lame example, as `vim` isn’t a very important process. In the real world, you’d be prioritizing processes that are actually important. In my case, since the `vim` process has a `PID` of `1789`, the command I would need to run in order to change the niceness would become this:

```
renice -n 10 -p 1789
```

The output of this command will appear as follows:



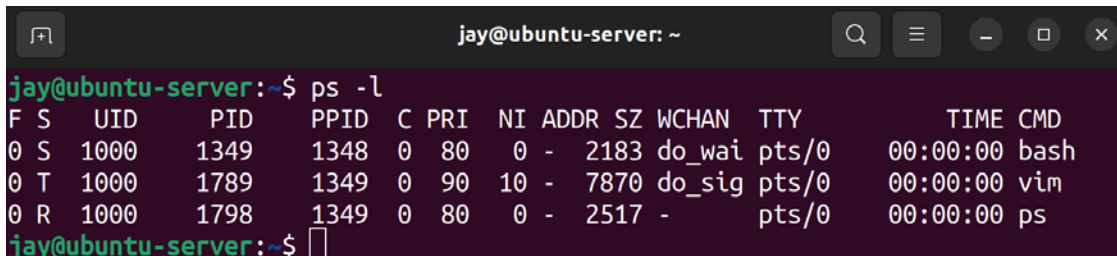
```

jay@ubuntu-server:~$ renice -n 10 -p 1789
1789 (process ID) old priority 0, new priority 10
jay@ubuntu-server:~$

```

Figure 7.6: Changing the priority of a process with `renice`

If we run `ps -l` again, we can see the new nice value for vim:



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ ps -l
F S  UID      PID     PPID  C  PRI  NI ADDR SZ WCHAN  TTY          TIME CMD
0 S  1000    1349    1348  0   80   0  - 2183 do_wai pts/0    00:00:00 bash
0 T  1000    1789    1349  0   90  10  - 7870 do_sig pts/0    00:00:00 vim
0 R  1000    1798    1349  0   80   0  - 2517 -      pts/0    00:00:00 ps
jay@ubuntu-server:~$

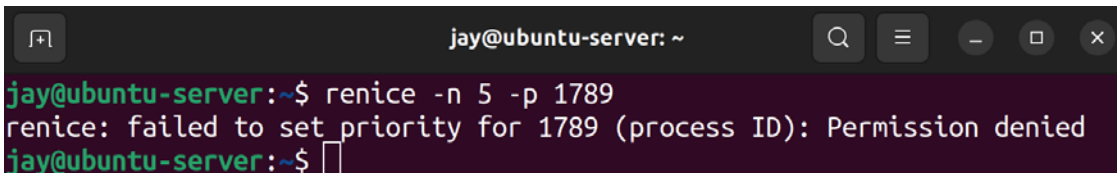
```

Figure 7.7: The output of the `ps -l` command after changing the priority of a process

The new nice value of 10 now shows up for vim under NI, and the PRI value has increased to 90. Now, this instance of vim will run at a lower priority than my other tasks, the reason being that the higher the nice value, the lower the priority. Notice that I didn't use sudo with the command when I changed the priority. In this example, that's okay because I'm increasing the nice value of the process, and that's allowed. However, let's try to decrease the nice value without sudo, using the following command:

```
renice -n 5 -p 1789
```

As you can see in the following output, I won't be as successful:



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ renice -n 5 -p 1789
renice: failed to set priority for 1789 (process ID): Permission denied
jay@ubuntu-server:~$

```

Figure 7.8: Attempting to decrease the priority of a process

My attempt to decrease the nice value from 10 down to 5 was blocked. If I were able to lower the niceness, then my process would be running at a higher priority. Instead, I received a Permission denied error. So essentially, users are allowed to increase the niceness of their processes, but are not allowed to decrease it, not even for processes they've initiated themselves. If you wish to decrease the nice value, you'll need to do so with sudo. So essentially, if you want to be "nicer," you can go ahead and do so. If you wish to be "meaner," you'll need root privileges. In addition, a user won't be able to change the priority of a process they don't own. So, if you attempt to use renice to change the niceness of a task running as a different user, you'll receive an Operation not permitted error.

At this point, we know how to re-prioritize our running processes with renice. Now, let's take a look at starting a new process with a specific priority with nice. Consider the following command:

```
nice -n 10 vim
```

Here, we're launching a new instance of `vim`, but with the priority set to a specific value right from the start. If we want to change the priority of `vim` again later, we'll need to use `renice`. As I mentioned earlier, `nice` is used to launch a new process with a specific priority, and `renice` is for changing the priority of a pre-existing process. In this example, we launched `vim` and set its `nice` value to 10 in one command.

Changing the priority of a text editor such as `vim` may seem like an odd choice for a test case, and it is. But the `vim` editor is harmless, as the likelihood of us changing the priority of it leading to a system halt is extremely minimal. There's no practical reason I can think of where it would be useful to re-prioritize something like a text editor. The takeaway, though, is that you *can* change the priority of the processes running on your server. On a real server, you may have an important process that runs and generates a report, and that report must be delivered on time. Or perhaps you have a process that generates an export of data that a client needs to have in order to make an on-time deliverable. So, if you think of the bigger picture, you can replace `vim` with the name of a process that is actually important for you or your organization.

You might be wondering what "nice" means in the context of the `nice` and `renice` commands. The "nice" number essentially refers to how nice a process is to other users. The higher the nice value, the lower the priority. So, a value of 20 is nicer than a value of 10. In that case, processes with a niceness of 20 are running at a lower priority, and so are kinder to the other processes on the system. The niceness can range from -20 to 19. A process with a nice value of -20 is the highest priority possible, while 19 is the lowest priority it can have. The entire system is quite a bit more complicated than this simple description. Although I refer to the nice value as the priority, it actually isn't. The nice value is used to calculate the actual priority. But for now, if we simplify the nice value to be representative of the priority, and the nice value to equate to a lower priority, the higher the number gets, that's enough for now.

So far, we've been using the `nice` and `renice` commands along with the `-n` option to set the nice values directly. It may be interesting to note though that you can simplify the `renice` command and leave out the `-n` option:

```
renice 10 42467
```

That command sets the nice value of the process to a positive 10, similar to our other examples. We can also use a negative number for the niceness if we want to increase the priority:

```
sudo renice -10 42467
```

Although it doesn't save us much typing to leave out the `-n` option, now you know that it is a possibility. The other difference with that example was that I needed to use `sudo` since I'm decreasing the nice value (more on that later).

When it comes to the `nice` command, we can also leave out the `-n` option, but the command works a bit differently in this regard. The following won't work:

```
nice 15 vim
```

The syntax of `nice` is a bit different, so giving it a positive number directly won't work as it does with `renice`. For that, we'll need to add a hyphen in front:

```
nice -15 vim
```

When you look at that command, you may assume we're applying a negative number. Actually, that's not true. Since the syntax is different with `nice`, the `-15` value we used results in a positive 15. We needed the hyphen in front of the value to signify to `nice` that we're applying a value as an option. If we actually do want to use a negative value with `nice` while also avoiding the `-n` option, we would need to use two hyphens:

```
nice --10 vim
```

The difference in syntax between the two commands with the `-n` option is a bit confusing in my opinion, so I recommend simply using the `-n` option with `nice` and `renice`, as that's going to be more uniform between them:

```
nice -n 10 vim
sudo nice -n -10 vim
renice -n 10 42467
sudo renice -n -10 42467
```

Those examples show both `nice` and `renice` using the `-n` option, and setting both positive and negative values. Since the `-n` option is used the same way between the two commands, it may be easier to focus on committing that to memory rather than focusing on the specifics. As previously discussed, I used `sudo` with commands that set a negative value for niceness, since only `root` can change a process to, or start a process with, a niceness below 0. You'll receive the following error if you try to do it anyway:

```
nice: cannot set niceness: Permission denied
```

This type of protection is somewhat important, because you may have some users who feel as though their processes are the most important, and try to prioritize them all the way to `-19`. At the end of the day, it's better for a system administrator to make decisions on which processes are allowed to reach a niceness value in the negative.

As an administrator of Ubuntu servers, it's up to you to decide which processes should be running, and at what priority. You'll then determine the best way to achieve the exact system state that's appropriate, and tuning process priority may be a part of that. If nothing else, learning the `nice` and `renice` commands gives you another utility for your toolset.

Dealing with misbehaving processes

Regarding the `ps` command, by this point you know how to display processes running on your server, as well as how to narrow down the output by string or resource usage. But what can you actually do with that knowledge? As much as we hate to admit it, sometimes the processes our server runs fail or misbehave and you need to restart them. If a process refuses to close normally, you may need to kill that process. In this section, we introduce the `kill` and `killall` commands to serve that purpose.

The `kill` command accepts a PID as an argument and attempts to close a process gracefully. In a typical workflow where you need to terminate a process that won't do so on its own, you will first use the `ps` command to find the PID of the culprit. Then, knowing the PID, you can attempt to `kill` the process. For example, if PID 31258 needed to be killed, you could execute the following:

```
sudo kill 31258
```

If all goes well, the process will end. You can restart it or investigate why it failed by perusing its logs.

To better understand what the `kill` command does, you first will need to understand the basics of **Linux signals**. Signals are used by both administrators and developers and can be sent to a process either by the kernel, another process, or manually with a command. A signal instructs the process of a request or change, and in some cases, to completely terminate. An example of such a signal is `SIGHUP`, which tells processes that their controlling terminal has exited. One situation in which this may occur is when you have a terminal emulator open, with several processes inside it running. If you close the terminal window (without stopping the processes you were running), they'll be sent the `SIGHUP` signal, which basically tells them to quit (essentially, it means the shell quit or hung up).

Other examples include `SIGINT` (where an application is running in the foreground and is stopped by pressing `Ctrl + c` on the keyboard) and `SIGTERM`, which, when sent to a process, asks it to cleanly terminate. Yet another example is `SIGKILL`, which forces a process to terminate uncleanly. In addition to a name, each signal is also represented by a value, such as 15 for `SIGTERM` and 9 for `SIGKILL`. Going over each of the signals is beyond the scope of this chapter (the advanced topics of signals are mainly only useful for developers), but you can view more information about them by consulting the man page if you're curious:

```
man 7 signal
```

For the purposes of this section, the two types of signals we are most concerned about are `SIGTERM`(15) and `SIGKILL`(9). When we want to stop a process, we send one of these signals to it, and the `kill` command allows us to do just that. By default, the `kill` command sends signal 15 (`SIGTERM`), which tells the process to cleanly terminate. If successful, the process will free its memory and gracefully close. With our previous example `kill` command, we sent signal 15 to the process, since we didn't clarify which signal to send.

Terminating a process with `SIGKILL`(9) is considered an extreme last resort. When you send signal 9 to a process, it's the equivalent of ripping the carpet out from underneath it or blowing it up with a stick of dynamite. The process will be force-closed without giving it any time to react at all, so it's one of those things you should avoid using unless you've literally tried absolutely everything you can think of. In theory, sending signal 9 can cause corrupted files, memory issues, or other shenanigans to occur. As for me, I've never actually run into long-term damage to software from using it, but theoretically it can happen, so you want to only use it in extreme cases. One case where such a signal may be necessary is regarding `defunct` or a **zombie process** in a situation where they don't close on their own. These processes are basically dead already and are typically waiting on their parent processes to reap them.

If the parent process never attempts to do so, they will remain on the process list. This in and of itself may not really be a big issue, since these processes aren't technically doing anything. But if their presence is causing problems and you can't kill them, you could try to send SIGKILL to the process. There should be no harm in eliminating a zombie process, but you would want to give them time to be reaped first.

To send signal 9 to a process, you would use the `-9` option of the `kill` command. It should go without saying, though, to make sure you're executing it against the proper process ID:

```
sudo kill -9 31258
```

Just like that, the process with a PID of 31258 will vanish without a trace. Anything it was writing to will be in limbo, and it will be removed from memory instantly. If, for some reason, the process still manages to stay running (which is extremely rare), you probably would need to reboot the server to get rid of it, which is something I've only seen in a few, very rare cases. An example of this is a zombie process, which is a process that shows up in the process list but isn't impacted by having signals sent to it, since such a process won't be scheduled for CPU time anyway. When it all comes down to it, if `kill -9` doesn't get rid of the process, nothing will.

Another method of killing a process is with the `killall` command, which is probably safer than the `kill` command (if for no other reason than there's a smaller chance you'll accidentally kill the wrong process). Like `kill`, `killall` allows you to send SIGTERM to a process, but unlike `kill`, you can do so by name. In addition, `killall` doesn't just kill one process, it kills any process it finds with the name you've given it as an option. To use `killall`, you would simply execute `killall` along with the name of a process:

```
sudo killall myprocess
```

Just like the `kill` command, you can also send signal 9 to the process as well:

```
sudo killall -9 myprocess
```

Again, use that only when necessary. In practice, though, you probably won't use `killall -9` very often (if ever), because it's rare for multiple processes under the same process name to become locked. If you need to send signal 9, stick to the `kill` command if you can.

The `kill` and `killall` commands can be incredibly useful in the situation of a stuck process, but these are commands you would hope you don't have to use very often. Stuck processes can occur in situations where applications encounter a situation from which they can't recover, so if you constantly find yourself needing to kill processes, you may want to check for an update to the package responsible for the service or check your server for hardware issues.

In the next section, let's take a look at system processes that run in the background and provide a service to us or our users, such as a web server process or DHCP server.

Managing system processes

System processes, also known as **daemons**, are programs that run in the background on your server and are typically started automatically when it boots. We don't usually manage these services directly as they run in the background to perform their duty, with or without needing our input. For example, if our server is a DHCP server and runs the `isc-dhcp-server` process, this process will run in the background, listening for DHCP requests and providing new IP assignments to them as they come in. Most of the time, when we install an application that runs as a service, Ubuntu will configure it to start when we boot our server, so we don't have to start it ourselves. Assuming the service doesn't run into an issue, it will happily continue performing its job forever until we tell it to stop. In Linux, services are managed by its `init` system, also referred to as PID 1 since the `init` system of a Linux system always receives that PID. In recent years, the way in which processes are managed in Ubuntu Server has changed considerably. Ubuntu has switched to `systemd` for its `init` system, which was previously `Upstart` until a few years ago. Ubuntu 16.04 was the first LTS release of Ubuntu with `systemd`, and this continues to be used today in Ubuntu 22.04. Since `systemd` has been the standard for quite some time now, we'll focus our attention on the commands used with it to manage our services. Older `init` systems are aging out.

With `systemd`, services are known as **units**, but for all intents and purposes, the terms “service,” “daemon,” and “unit” all essentially mean the same thing. Since I started using Linux over 20 years ago, I still refer to `systemd` units as services, out of habit. To help us manage these “units,” `systemd` includes the `systemctl` command, which allows you to start, stop, and view the status of units on our server. To help illustrate this, I'll use OpenSSH as an example. The name of the unit doesn't really matter, as the syntax of the `systemctl` command is the same regardless of the name of the unit we're interacting with. You can use `systemctl` to start, stop, or restart your Apache instance, your database server, or even use it to restart the entire networking stack. The `systemctl` command, with no options or parameters, assumes the `list-units` option, which dumps a list of units to your shell. This can be a bit messy, though, so if you already know the name of a unit you'd like to search for, you can pipe the output into `grep` and search for a string. This is handy in a situation where you may not know the exact name of the unit, but you know part of it:

```
systemctl | grep ssh
```

If you want to check the health of a unit, the best way is to actually use the `status` keyword, which will show you some very useful information regarding the unit. This information includes whether or not the unit is running, if it's enabled (meaning it's configured to start at boot time), as well as the most recent log entries for the unit:

```
systemctl status ssh
```


This command will produce an output something like the following:

```

jay@ubuntu-server: ~
● ssh.service - OpenBSD Secure Shell server
   Loaded: loaded (/lib/systemd/system/ssh.service; enabled; vendor preset: enabled)
   Active: active (running) since Mon 2022-05-09 01:06:46 UTC; 2h 4min ago
     Docs: man:sshd(8)
           man:sshd_config(5)
   Process: 736 ExecStartPre=/usr/sbin/sshd -t (code=exited, status=0/SUCCESS)
  Main PID: 768 (sshd)
    Tasks: 1 (limit: 2241)
   Memory: 7.6M
      CPU: 90ms
   CGroup: /system.slice/ssh.service
           └─768 "sshd: /usr/sbin/sshd -D [listener] 0 of 10-100 startups"

May 09 01:06:46 ubuntu-server systemd[1]: Starting OpenBSD Secure Shell server...
May 09 01:06:46 ubuntu-server sshd[768]: Server listening on 0.0.0.0 port 22.
May 09 01:06:46 ubuntu-server systemd[1]: Started OpenBSD Secure Shell server.
lines 1-16

```

Figure 7.9: Checking the status of a unit with `systemctl`

Most of the time, you can actually check the status of units without needing root access, but you may not see all the information available. In the screenshot, you can see several log entries for the `ssh` service, but some units do not show those entries without `sudo`. With the `ssh` unit in particular, we see the log entries when checking the status with or without `sudo`.

Another thing you may notice in the screenshot is that the name of the `ssh` unit is actually `ssh.service`, but you don't need to include the `.service` part of the name, since that is implied by default. Sometimes, while viewing the status of a process with `systemctl`, the output may be condensed to save space on the screen. To avoid this and see the full log entries, add the `-l` option:

```
systemctl status -l ssh
```

Another thing to pay attention to is the `vendor preset` of the unit. Most packages in Ubuntu that include a service file for `systemd` will enable it automatically, but other distributions typically don't start and enable units by default (such as CentOS). In the case of the `ssh` example, you can see that the `vendor preset` is set to `enabled`. This means that once you install the `openssh-server` package, the `ssh.service` unit will automatically be enabled. You can confirm this by checking the `Active` line (where the example output says `active (running)`), which tells us that the unit is running. The `Loaded` line clarifies that the unit is enabled, so we know that the next time we start the server, `ssh` will be loaded automatically. Although `systemd` units are typically enabled and started automatically in Ubuntu when installing their package, this can still vary. When you install a new package, make sure you check the status of the unit, so you'll be aware of its settings.

Starting and stopping a unit is just as easy; all you have to do is change the keyword you use with `systemctl` to start or stop in order to have the desired effect:

```
sudo systemctl stop ssh
sudo systemctl start ssh
```

There are additional keywords, such as `restart` (which takes care of the previous two command examples at the same time), and some units even feature `reload`, which allows you to activate new configuration settings without needing to bring down the entire application. An example of why this is useful is with Apache, which serves web pages to local or external users. If you stop Apache, all users will be disconnected from the website you're serving. If you add a new site, you can use `reload` rather than `restart`, which will activate any new configuration you may have added without disturbing the existing connections. We'll take a look at Apache in *Chapter 14, Serving Web Content*, so don't worry too much about Apache right now. It's just a good example of a unit with additional functionality. Not all units feature a `reload` option, so you should check the documentation of the application that provides the unit to be sure.

Since I mentioned starting and stopping the unit for OpenSSH in the previous examples, an interesting aside is that doing so will not disconnect a current SSH session to the server, should you have one open. If you stop the `ssh` service, it won't drop your connection. Open connections are maintained, and stopping SSH only prevents new connections from happening. Therefore, SSH is different when compared to other units (such as Apache) in that existing connections aren't dropped when restarting the unit.

As I mentioned before, if you want a unit to automatically start when the server boots, the unit will need to be enabled. Units are automatically enabled most of the time, but in case you find one that isn't enabled, you can enable it with the `enable` keyword:

```
sudo systemctl enable ssh
```

It's just as easy to disable a unit as well:

```
sudo systemctl disable ssh
```

You can combine the process of not only enabling a unit, but also starting it at the same time:

```
sudo systemctl enable --now ssh
```

The `--now` argument tells `systemctl` to start the unit immediately after enabling it, rather than waiting for the next boot to do so or having you also run the `start` argument in a separate command.

Even though `systemd` is primarily used for managing units, it's actually an entire platform that manages multiple things on a Linux system, including DNS resolving, networking, and more. `systemd` even handles logging as well, and it provides us with the `journalctl` command, which we can use to view logging info (this is also why the output of `systemctl status ssh` was able to show us log entries).

We've discussed logging a bit in *Chapter 4, Navigating and Essential Commands*, and we'll do so in more detail during *Chapter 22, Troubleshooting Ubuntu Servers* (which will also include further discussion of the `journalctl` command).

For now, just understand that `systemd` is quite extensive when it comes to the number of things it helps us manage. For the purposes of this chapter, however, if you understand how to start, stop, enable, disable, and check the status of a unit, you're good to go for now.

Scheduling tasks with cron

Earlier in this chapter, we worked through starting processes and enabling them to run in the background, and ensuring they start as soon as the server boots. In some cases, you may need an application to perform a job at a specific time, rather than to have it always running in the background. This is where `cron` comes in. With `cron`, you can set a process, program, or script to run at a specific time, down to the minute. Each user is able to have their own set of `cron` configurations (known as a `crontab`), which can perform any function that a user would be able to do normally. The `root` user has a `crontab` as well, which allows system-wide administrative tasks to be performed. Each `crontab` includes a list of `cron` jobs (one per line), which we'll get into shortly. To view a `crontab` for a user, we can use the `crontab` command:

```
crontab -l
```

With the `-l` option, the `crontab` command will show you a list of jobs for the user who executed the command. If you execute it as `root`, you'll see the `root` account's `crontab`. If you execute it as user `jdoe`, you'll see the `crontab` for `jdoe`, and so on. If you want to view a `crontab` for a user other than yourself, you can use the `-u` option and specify a user, but you'll need to execute it as `root` or with `sudo` to view the `crontab` for someone other than the user you're logged in as:

```
sudo crontab -u jdoe -l
```

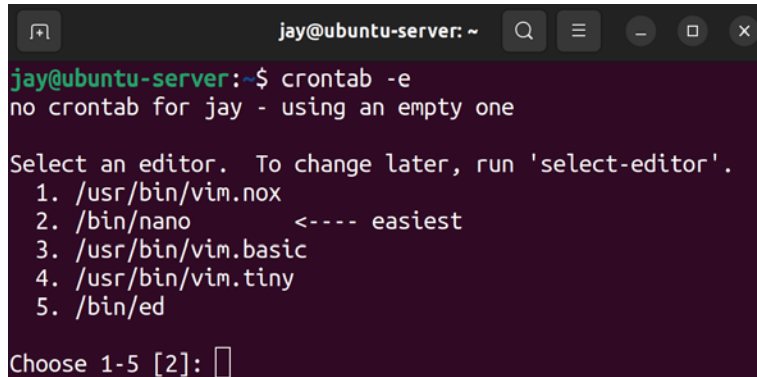
By default, no user has a `crontab` until you create one or more jobs. Therefore, you'll probably see output such as the following when you check for your current users:

```
no crontab for jdoe
```

To create a `cron` job, first log in as the user account you want the task to run under. Then, issue the following command:

```
crontab -e
```

If you have more than one text editor on your system, you may see output similar to the following:



```

jay@ubuntu-server:~$ crontab -e
no crontab for jay - using an empty one

Select an editor. To change later, run 'select-editor'.
 1. /usr/bin/vim.nox
 2. /bin/nano          <---- easiest
 3. /usr/bin/vim.basic
 4. /usr/bin/vim.tiny
 5. /bin/ed

Choose 1-5 [2]: 2

```

Figure 7.10: Selecting an editor for use with the crontab command

In this case, you'll simply press the number corresponding to the text editor you'd like to use when creating your cron job. To set an environment variable that specifies a specific editor and edit your crontab with a single command, the following command will do exactly that:

```
EDITOR=vim crontab -e
```

In this example, you can replace `vim` with whatever text editor you prefer. At this point, you should be placed in a text editor with your crontab file open. The default crontab file for each user features some helpful comments that give you some useful information regarding how cron works. To add a new job, you would scroll to the bottom of the file (after all the comments) and insert a new line. Formatting is very particular here, and the example comments in the file give you some clue as to how each line is laid out. Specifically, this part:

```
m h dom mon dow command
```

Each cron job has six fields, each separated by at least one space or tab spaces. If you use more than one space, or tab, cron is smart enough to parse the file properly. In the first field, we have the minute in which we would like the job to occur. In the second field, we place the hour in the 24-hour format, from 0-23. The third field represents the day of the month. In this field, you can place a 5 (5th of the month), 23 (23rd of the month), and so on. The fourth field corresponds to the month, such as 3 for March or 12 for December. The fifth field is the day of the week, numbered from 0 to 6 to represent Sunday through Saturday. Finally, in the last field, we have the command to be executed. A few example crontab lines are as follows:

```
3 0 * * 4 /usr/local/bin/cleanup.sh
* 0 * * * /usr/bin/apt update
0 1 1 * * /usr/local/bin/run_report.sh
```

With the first example, the `cleanup.sh` script, located in `/usr/local/bin`, will be run at 12:03 a.m. every Thursday. We know this because the minute column is set to 3, the hour column is set to 0 (midnight), the day column is 4 (Thursday), and the command column shows a fully qualified command of `/usr/local/bin/cleanup.sh`.

What does it mean for a command to be *fully qualified*? Basically, a command being fully qualified means that the entire path to the binary responsible for the command is completely typed out. In the second example, we could have simply typed `apt update` for the command and that would've probably worked just fine. However, not including the full path to the program is considered bad cron etiquette. While the command may have worked without being fully qualified, its success would depend on the application being found in the path of the user who is calling it. Not all servers are set up the same, so this might not work depending on how the shell is set up. If you include the full path, the job should run regardless of how the underlying shell is configured.

If you don't know what the fully qualified command is, all you have to do is use the `which` command. This command, when used with the name of a command you'd like to run, will give you the fully qualified command if the command is located on your system.

Continuing with the second example, we're running `/usr/bin/apt update` to update our server's repository index every morning at midnight. The asterisks on each line refer to *any*, so with the minute column being simply `*`, that means that this task is eligible for any minute. Basically, the only field we clarified was the hour field, which we set to `0` in order to represent 12:00 a.m.

With the third example, we're running the `/usr/local/bin/run_report.sh` script on the first day of every month at 01:00 a.m. If you notice, we set the third column (**day of month**) to `1`, which is the same as February 1st, March 1st, and so on. This job will be run if it's the first day of the month, but only if the current time is also 01:00 a.m., since we filled in the first and second columns, which represent the minute and hour, respectively.

Once you finish editing a user's crontab and save it, `cron` is updated and, from that point forward, will execute the task at the time you select. The crontab will be executed according to the current time and date on your server, so you want to make sure that that is correct as well, otherwise you'll have your jobs execute at unexpected times. You can view the current date and time on your server by simply issuing the `date` command.

To get the hang of creating jobs with `cron`, the best way (as always) is to practice. The second example cron job is probably a good one to experiment with, as updating your repository index isn't going to hurt anything.

Summary

In this chapter, we learned how to manage processes. We began with a look at the `ps` command, which we can use to view a list of processes that are currently running. We also took a look at managing jobs, as well as killing processes that, for one reason or another, are misbehaving. We also discussed methods of changing the priority of a process, to ensure we have full control over which processes are given more processing time, and we also learned how we can schedule things to run at a later time and date with `cron`.

In *Chapter 8, Monitoring System Resources*, we'll take a look at some ways we can keep an eye on the resources that are available on our server, where we will learn how to check disk usage and understand memory usage and swap space, as well as a looking at some utilities that can make resource management a breeze.

Relevant videos

- Getting started with tmux (LearnLinuxTV): <https://linux.video/tmux-guide>
- Backgrounding and foregrounding processes in Linux (LearnLinuxTV): <https://linux.video/bg-fg>

Further reading

- Ham Vocke, *A Quick and Easy Guide to tmux*: <https://learnlinux.link/tmux-article>
- Tmux cheat sheet and quick reference: <https://learnlinux.link/tcs>
- crontab guru: <https://learnlinux.link/ctg>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaz0>



8

Monitoring System Resources

In the last chapter, we learned how we can manage tasks that are running on our server. We now know how to see what's running in the background, how to enable or disable a unit from starting at boot time, and also how to schedule tasks to run in the future. But in order for us to be able to effectively manage the tasks that our servers carry out, we also need to keep an eye on system resources. If we run out of RAM, fill up our disk, or overload our CPU, then a server that normally processes tasks very efficiently might come to a screeching halt. In this chapter, we'll take a look at these resources and how to monitor them.

Our discussion on resource management will include:

- Viewing disk usage
- Monitoring memory usage
- Understanding load average
- Viewing resource usage with `htop`

One resource that is extremely important on our servers is storage, and keeping track of such things as available disk space is critical – even the most powerful server you can purchase would be unable to function without free disk space. We'll take a look at some ways to monitor disk usage in the next section.

Viewing disk usage

Keeping an eye on your storage is always important, as no one enjoys getting a call in the middle of the night saying that a server encountered an issue, especially not something that could've been easily avoided, such as a filesystem growing too close to being full. Managing storage on Linux systems is simple once you master the related tools, the most useful of which I'll go over in this section. In particular, we'll look at tools we can use to answer the question “what's using up all the disk space?”, which is the most common question that comes up when dealing with disk usage.

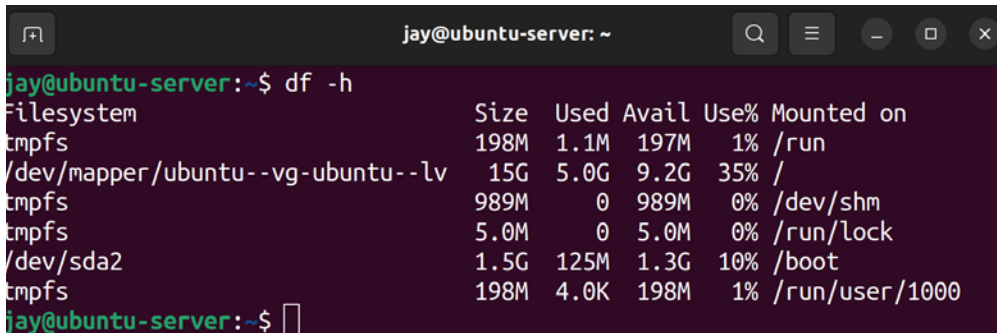
First, let's look at the `df` command.

Using df

The `df` command is likely always going to be your starting point in situations where you don't already know which volume or mount point is becoming full. When executed, it gives you a high-level overview, so it's not necessarily useful when you want to figure out who or what in particular is hogging all your space. However, when you just want to list all your mounted volumes and see how much space is left on each, `df` fits the bill. By default, it shows you the information in bytes. However, I find it easier to use the `-h` option with `df`, which will show output that's more human-readable, and by doing so you'll see information that's a bit easier to read. Go ahead and give it a try:

```
df -h
```

This should produce an output that looks something like the following:



```
jay@ubuntu-server: ~  
jay@ubuntu-server:~$ df -h  
Filesystem                Size      Used Avail Use% Mounted on  
tmpfs                    198M    1.1M  197M   1% /run  
/dev/mapper/ubuntu--vg-ubuntu--lv  15G    5.0G   9.2G  35% /  
tmpfs                    989M         0  989M   0% /dev/shm  
tmpfs                    5.0M         0   5.0M   0% /run/lock  
/dev/sda2                 1.5G    125M   1.3G  10% /boot  
tmpfs                    198M     4.0K   198M   1% /run/user/1000  
jay@ubuntu-server:~$
```

Figure 8.1: Output from the `df -h` command

The output will look different depending on the types of disks and mount points associated with your system. In the screenshot, you'll see that the root filesystem is located on `/dev/mapper/ubuntu--vg-ubuntu--lv`. We know this because under the column `Mounted on` we see that the mount point is set to a single forward slash (`/`). As we discussed in *Chapter 4, Navigating and Essential Commands*, this single forward slash refers to the beginning of the filesystem (also referred to as the root filesystem). In my case, this is an LVM volume, which is why we have a device with such a long name, beginning with `/dev/mapper`. Let's not worry about LVM at this point – we'll discuss that later. But for now, just keep in mind that the single forward slash refers to the beginning of the filesystem, and the device name on the left refers to the actual device that's mounted there.

The actual device name varies from one server to another and also varies depending on whether you chose to utilize LVM during installation. Instead of a long path beginning with `/dev/mapper`, you may instead see the device name as `/dev/sda1`, `/dev/xvda1`, `/dev/nvme0n1p1`, or other variations. The name of the device is generated by the type of hardware the underlying storage device is, such as the `/dev/nvme...` naming convention used for NVME hard drives, `/dev/sdaN` for standard SATA hard drives, and so on.

The actual type of device the underlying storage hardware is doesn't matter so much; it only really matters that you can identify which device is in the most danger of becoming full. In the example screenshot, the root filesystem is using 35% of its available space. In this case, we aren't in danger of running out of space when it comes to that.

If you do see that an important storage volume is either full or trending toward becoming full, then you'll know which one in particular you should focus on, and we'll explore additional ways that you can obtain more information about what's using up space very shortly.

However, sometimes a storage volume can be considered full, even when it appears that the volume has plenty of space free. This is due to the fact that on a Linux system, the actual data you're storing and the size of that data isn't the only consideration. We also have to take inodes into account as well.

But, what exactly is an inode, and why would such a thing cause a disk to be reported as full when it's actually not? Think of the concept of an inode as a type of *database object*, containing metadata for the actual items you're storing. Information stored in inodes are details such as the owner of the file, permissions, last modified date, and type (whether it is a directory or a file). While metadata is certainly a good thing to have, the problem with inodes is that you can only have a limited number of them on any storage device. If a storage device reaches its inode limit, then the volume is still considered full, and will not be able to accept additional data.

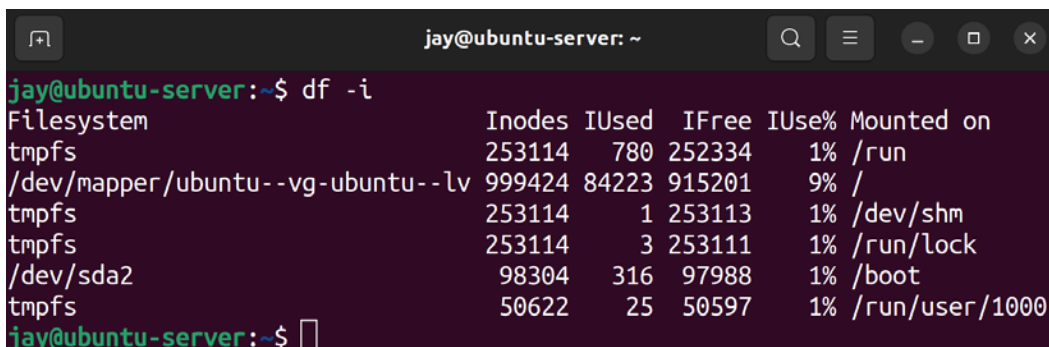
In practice, the symptom of this scenario is that commands such as `df` will show the volume as having free space, yet when you try to save a new file onto the device, you will see an error that you're unable to do so because the volume is full. If you weren't aware of the existence of inodes, then this situation might be a bit confusing.

While it may seem as though having another limit on storage in the form of inodes is a downside, in reality, the inode limit on storage volumes is usually extremely high and very hard to reach. Often, if an inode limit is reached, that usually means that there's a bigger issue with the server that's causing it to hit this limit. For example, perhaps there's an issue on the server where it's saving a much higher number of files than it should, such as an usual number of log files or queued e-mail message files.

Thankfully, determining whether or not you're running out of inodes on a particular storage volume is very simple – rather than using the `-h` option with the `df` command, use the `-i` option instead. The `-i` option will display inode counts instead of standard size-based storage metrics. To help illustrate the difference, I'll show you some output from one of my servers to give you an idea of what this looks like:

```
df -i
```

The output of that command on my system is as follows:



```
jay@ubuntu-server: ~  
jay@ubuntu-server:~$ df -i  
Filesystem          Inodes IUsed  IFree IUse% Mounted on  
tmpfs                253114   780 252334    1% /run  
/dev/mapper/ubuntu--vg-ubuntu--lv 999424 84223 915201    9% /  
tmpfs                253114     1 253113    1% /dev/shm  
tmpfs                253114     3 253111    1% /run/lock  
/dev/sda2            98304   316 97988    1% /boot  
tmpfs                50622    25 50597    1% /run/user/1000  
jay@ubuntu-server:~$
```

Figure 8.2: Output from the `df -i` command

In this example, the root filesystem on the example server has a total of 999424 inodes available, of which 84223 are used and 915201 are free. In my case, I have plenty of inodes available. However, I recommend committing the `df -h` and `df -i` commands to memory. Whether any space issues with storage you may experience pertain to actual space or inode utilization, you'll be able to know which is the case between those two commands.

Assuming you have storage that's on the verge of becoming full (or it already is), how do you pinpoint exactly what in particular is using up all that space? There are additional tools you can use that will help you narrow this down. And that's exactly what we'll explore next.

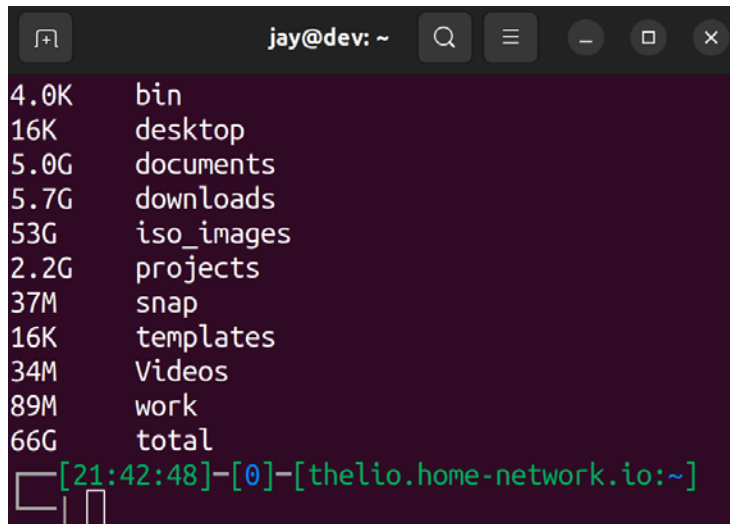
Diving deeper into disk usage

The next step in investigating what's gobbling up your disk space is finding out which files in particular are using it all up. At this stage, there is a multitude of tools you can use to investigate. The first I'll mention is the `du` command, which is able to show you how much space a directory is using. Using `du` against directories and sub-directories will help you narrow down the problem. Like `df`, we can also use the `-h` option with `du` to make our output easier to read. By default, `du` will scan the current working directory your shell is attached to and give you a list of each item within the directory, the total space each item consists of, as well as a summary at the end.

The `du` command is only able to scan directories that its calling user has permission to scan. If you run this as a non-root user, then you may not be getting the full picture. Also, the more files and sub-directories that are within your current working directory, the longer this command will take to execute. If you have an idea where the resource hog might be, try to `cd` into a directory further in the filesystem tree to narrow your search down and reduce the amount of time the command will take. The output of `du -h` can often be more verbose than you actually need in order to pinpoint your culprit and can fill several screens. To simplify it, my favorite variation of this command is the following:

```
du -hsc *
```

Basically, you would run `du -hsc *` within a directory that's as close as possible to where you think the problem is. The `-h` option, as we know, gives us human-readable output (essentially, giving us output in the form of megabytes, gigabytes, and so on). The `-s` option gives us a summary and `-c` provides us with the total amount of space used within our current working directory. The following screenshot shows this output from one of my computers:



```
jay@dev: ~
4.0K  bin
16K   desktop
5.0G  documents
5.7G  downloads
53G   iso_images
2.2G  projects
37M   snap
16K   templates
34M   Videos
89M   work
66G   total
[21:42:48]-[0]-[thelio.home-network.io:~]
```

Figure 8.3: Example output from `du -hsc *`

To make that example more interesting, I took the screenshot from my personal desktop, but the resulting command and its syntax won't differ from one device to the next. As you can see, the information provided by `du -hsc *` is a nice, concise summary. From the output, we can clearly see how much space each of the directories within our working directory takes currently. For example, I have 2.2 GB used in my `projects` directory right now, as well as 53 GB worth of ISO images.

At this point, we know which directories at the top level of our current working directories are using the most space. But we still need to narrow this down to *what* in particular within those directories is responsible for using that space. To dive deeper, we could `cd` into any of those large directories and run the `du` command again. After a few runs, we should be able to narrow down the largest files within these directories and make a decision on what we want to do with them. Perhaps we can clean unnecessary files or add another disk. Once we know what is using up our space, we can decide what we're going to do about it.

At this point in reading this book, you're probably under the impression that I have some sort of strange fixation on saving the best for last. You'd be right. I'd like to finish off this section by introducing you to one of my favorite applications, the **NCurses Disk Usage** utility (or more simply, `ncdu`). The `ncdu` command is one of those things that administrators who constantly find themselves dealing with disk space issues learn to love and appreciate. In one go, this command gives you not only a summary of what is eating up all your space but also gives you the ability to traverse the results without having to run a command over and over while manually traversing your directory tree. You simply execute it once and then you can navigate the results and drill down as far as you need.

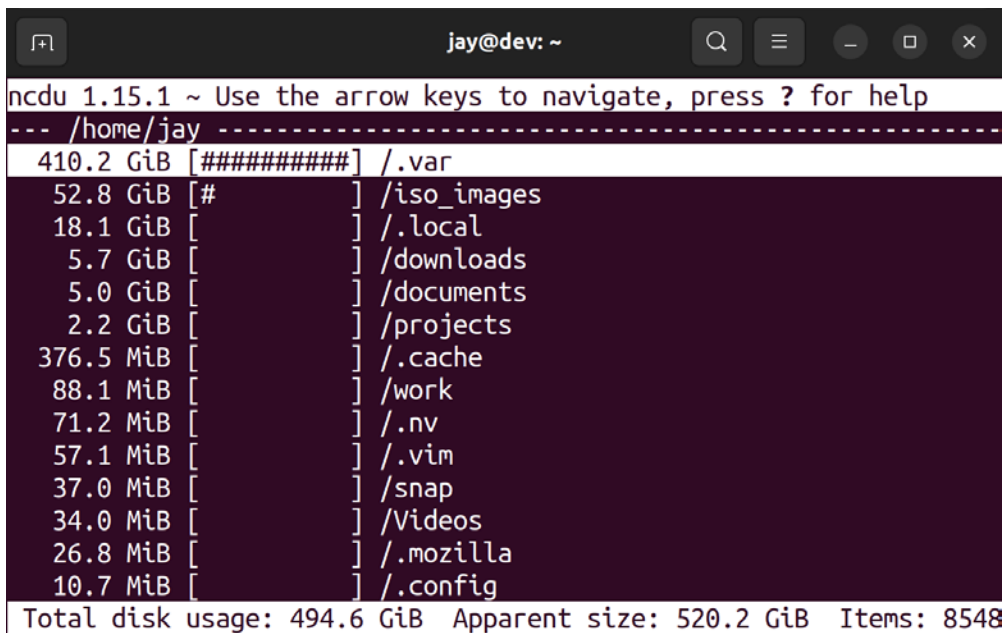
To use `ncdu`, you will need to install it as it doesn't come with Ubuntu by default:

```
sudo apt install ncdu
```

Once installed, simply execute `ncdu` in your shell from any starting directory of your choosing. When done, simply press `q` on your keyboard to quit. Like `du`, `ncdu` is only able to scan directories that the calling user has access to. You may need to run it as root to get an accurate portrayal of your disk usage.

You may want to consider using the `-x` option with `ncdu`. This option will limit it to the current filesystem, meaning it won't scan network mounts or additional storage devices; it'll just focus on the device you started the scan on. This can save you from wasting time scanning areas that aren't related to your issue.

When executed, `ncdu` will scan every directory from its starting point onward. When finished, it will give you a menu-driven layout allowing you to browse through your results:



```
jay@dev: ~  
ncdu 1.15.1 ~ Use the arrow keys to navigate, press ? for help  
--- /home/jay -----  
410.2 GiB [#####] /.var  
52.8 GiB [#          ] /iso_images  
18.1 GiB [           ] /.local  
5.7 GiB [            ] /downloads  
5.0 GiB [            ] /documents  
2.2 GiB [            ] /projects  
376.5 MiB [          ] /.cache  
88.1 MiB [           ] /work  
71.2 MiB [           ] /.nv  
57.1 MiB [           ] /.vim  
37.0 MiB [           ] /snap  
34.0 MiB [           ] /Videos  
26.8 MiB [           ] /.mozilla  
10.7 MiB [           ] /.config  
Total disk usage: 494.6 GiB Apparent size: 520.2 GiB Items: 8548
```

Figure 8.4: `ncdu` in action

Again, I took this screenshot from my desktop, from within my home directory. What `ncdu` does is show you your disk usage from your current directory down, and it will order the results by placing the items with the highest usage toward the top. To move around inside of `ncdu`, you do so by moving your selection (indicated with a long white highlight) with the up and down arrows on your keyboard.

If you press *Enter* on a directory, `ncdu` switches to showing you the summary of that directory, and you can continue to drill down as far as you need. In fact, you can actually delete items and entire folders by pressing *d*. Therefore, `ncdu` not only allows you to find what is using up your space but allows you to take action as well!

Sometimes, it's obvious what's taking up space on a disk, and `ncdu` may not always be necessary. Generally speaking, you'll start out your investigation with `df -h`, to see which storage volume is the one that's running out of space. Then, you'll go into that directory and run another command, such as `du -hsc *`, to see which directory is using up the most space. If you don't immediately know from the output of `du` what the underlying issue is, then consider using a tool such as `ncdu` to dive down even deeper.

Although monitoring storage is critical, we also need to keep an eye on free memory. Next up, we'll take a look at how to monitor the memory of our server.

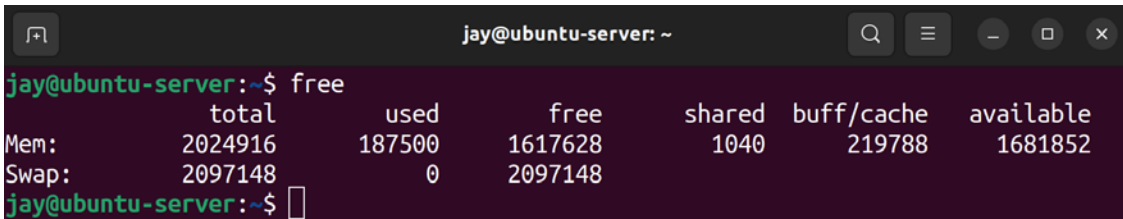
Monitoring memory usage

I forget things all the time. I regularly forget where my car keys are, even though they're almost always right there in my pocket the entire time. I even forget to use `sudo` for commands that normally require it, despite working with Linux for over 20 years. Thankfully, computers have a better memory than I do, but if we don't manage it effectively, the memory on our servers will be just as useless as I am when I forget to put freshly washed laundry in the dryer.

Understanding how Linux manages memory can actually be a somewhat complex topic, as understanding how much memory is truly free can be a hurdle for newcomers to overcome. You'll soon see that how Linux manages memory on your server is actually fairly straightforward once explained.

Understanding server memory

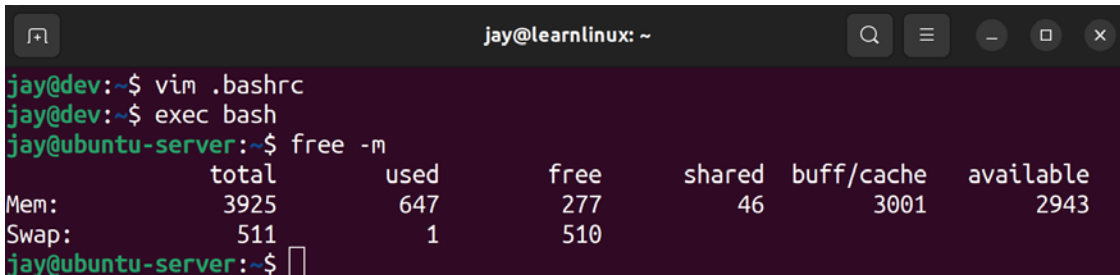
For the purpose of monitoring memory usage on our server, we have the `free` command at our disposal, which we can use to see how much memory is being consumed at any given time. Giving the `free` command with no options will result in the output being shown in terms of kilobytes:



```
jay@ubuntu-server: ~
jay@ubuntu-server:~$ free
              total        used        free      shared  buff/cache   available
Mem:           2024916          187500      1617628           1040         219788      1681852
Swap:          2097148              0          2097148
```

Figure 8.5: Output of the `free` command

My favorite variation of this command is `free -m`, which shows the amount of memory in use in terms of megabytes. You can also use `free -g` to show the output in terms of gigabytes, but the output won't be precise enough on most servers. In my opinion, adding the `-m` option makes the `free` command much more readable:



```

jay@dev:~$ vim .bashrc
jay@dev:~$ exec bash
jay@ubuntu-server:~$ free -m
              total        used         free       shared    buff/cache   available
Mem:           3925         647         277          46         3001       2943
Swap:           511           1         510
jay@ubuntu-server:~$

```

Figure 8.6: Output of the `free -m` command

Since everything is broken down into megabytes, it's much easier to read, at least for me.

At first glance, it may appear as though this server has only 277 MB free. You'll see this in the first row and third column under `free`. In actuality, the number you'll really want to pay attention to is the number under `available`, which is 2943 MB in this case. That's now much memory is actually free. Since this server has 3925 MB of total RAM available (you'll see this on the first row, under `total`), this means that most of the RAM is free, and this server is not really working that hard at all.

Some additional explanation is necessary to truly understand these numbers. You could very well stop reading this section right now as long as you take away from it that the `available` column represents how much memory is free for your applications to use. However, it's not quite that simple.

Technically, when you look at my output, the server really does have only 277 MB of memory free. The amount of memory listed under `available` is legitimately being used by the system in the form of a cache but would be freed up in the event that any application needed to use it. If an application starts and needs a decent chunk of memory in order to run, the kernel will provide it with some memory from this cache.

Linux, like most modern systems, subscribes to the belief that “unused RAM is wasted RAM.” RAM that isn't being used by any process is given to what is known as a **filesystem cache**, which is utilized to make your server run more efficiently. When data needs to be written to a storage device, it's not directly written right away. Instead, this data is written to the filesystem cache (a portion of RAM that's set aside) and then synchronized to the storage device later in the background. The reason this makes your server more efficient is that this data being stored in RAM would be written to and retrieved faster than it would be from disk. Applications and services can synchronize data to the disk in the background without forcing you to wait for it. This cache also works for reading data, as when you first open a file, its contents are cached. The system will then retrieve it from RAM if you read the same file again, which is more efficient than loading it from the storage volume each time. If you just recently saved a new file and retrieve it right away, it's likely still in the cache and then retrieved from there, rather than from the disk directly.

To understand all of the columns shown in *Figure 8.6*, I'll outline the meaning of each in the following table:

Column	Meaning
total	The total amount of memory installed on the server.
used	The memory that is used (from any source). This is calculated as follows: $used = total - free - buffers/cache$.
free	The memory not being used by anything, the cache or otherwise.
shared	The memory used by tmpfs as well as other shared resources.
buff/cache	The amount of memory being used by the buffers and cache.
available	The memory that is free for applications to use.

You may have noticed in *Figure 8.6* that memory usage is also listed for a resource called `swap`. Let's take a look at that as well. We will dedicate the next section entirely to it so that we ensure we understand what it is, and what it does for us.

Managing swap

`swap` is one of those things we never want to use, but always want to make sure it is available. It's kind of like car insurance, no one is excited to buy it, but we do want to have it in case something bad happens. There's even some debate between administrators on whether or not `swap` is still relevant today. It's definitely relevant, regardless of what anyone says, as it's a safety net of sorts. (And disk space is cheaper nowadays, so dedicating some of our storage to this task isn't really a big deal, so we may as well.)

So what is it? `swap` is basically a partition or a file that acts as RAM in situations where your server's memory is saturated. If we manage a server properly, we hope to never need it, as `swap` is stored on your hard disk, which is orders of magnitude slower than RAM. But if something goes wrong on your server and your memory usage skyrockets, `swap` may save you from having your server go down. The **Out of Memory (OOM) Killer** may also activate itself when memory is full, to kill a misbehaving process that's using the majority of your memory, but as much as possible, we don't want to rely on that and instead ensure adequate `swap` in case memory is exhausted.

The way `swap` is implemented by default in Ubuntu is in the form of a `swap` file. In previous versions of Ubuntu (specifically, 16.04 and earlier) it was implemented via a `swap` partition instead. In fact, if you have an existing server that you've upgraded to Ubuntu 22.04 and it started on an older Ubuntu version, you may still have a `swap` partition, even though you're running the latest release. New installations of Ubuntu performed after 16.04 will have a `swap` file.

Is having a `swap` file better than a `swap` partition? I would say yes, it's preferred – although you won't notice any difference when it comes to performance. Whether `swap` on your server is in the form of a `swap` file or partition, it doesn't change the fact that `swap` uses your disk and is slower than RAM. One benefit of a `swap` file compared to a `swap` partition is that it's easier to grow or shrink a `swap` file than it is to do the same with a `swap` partition.

Anyway, considering that swap files are the preferred method (and the new default) going forward, I won't cover the process of creating swap as a partition, as there's no reason to do so anymore.

The swap file for your server is declared in the `/etc/fstab` file (we'll discuss the `/etc/fstab` file in more detail in *Chapter 9, Managing Storage Volumes*). In most cases, you would've had a swap file created for you during installation. You could, of course, add a swap file later if for some reason you don't have one. In the case of some cloud instance providers, you may not get a swap file by default. In that situation, you would create a swap file yourself (we'll discuss the process later in this section) and then use the `swapon` command to activate it:

```
sudo swapon -a
```

When run, the `swapon -a` command will find your swap file in `/etc/fstab` (if one is listed there), mount it, and activate it for use. The inverse of this command is `swapoff -a`, which deactivates your swap file. It's rare that you'd need to disable swap, unless, of course, you were planning on deleting your swap file in order to create a larger one. If you find out that your server has inadequate swap, that may be a course of action you would take.

While having swap is generally a good idea, there are actually some applications that prefer that the server doesn't have it at all. Not having swap isn't a common requirement, but Kubernetes is a good example of a situation that might lead you to disabling swap altogether. In fact, the installation process for Kubernetes will complain (or possibly fail) if you do have swap enabled. In the case of a Kubernetes cluster, the individual servers within such a cluster would be a special case anyway, each dedicated to the task of running containers (which is what Kubernetes does; more on that in *Chapter 18, Container Orchestration*).

When you check your free memory (hint: execute `free -m`), you'll see swap listed whether you have it or not, but when swap is deactivated, you will see all zeros for the size totals.

So, how do you actually create a swap file if you wish to use it and you don't already have one? To do so, you'll first create the actual file to be used as swap. This can be stored anywhere, but `/swapfile` is typically ideal. You can use the `fallocate` command to create the actual file. The `fallocate` command will force a file to be a particular size:

```
sudo fallocate -l 4G /swapfile
```

Here, I'm creating a 4 GB swap file, but feel free to make yours whatever size you want in order to fit your needs. Next, we need to prepare this file to be used as swap. First, we'll need to fix the permissions as we need this file to be a bit more restrictive than most:

```
sudo chmod 0600 /swapfile
```

Then, we can use the `mkswap` command to convert this file into an actual swap file:

```
sudo mkswap /swapfile
```

Now, we have a handy-dandy swap file stored on our root filesystem. Next, we'll need to mount it. As always, it's recommended that we add this to our `/etc/fstab` file. What follows is an example entry:

```
/swapfile none swap sw 0 0
```

From this point, we can activate our new swap file with the `swapon` command that I mentioned earlier:

```
sudo swapon -a
```

After running that command, the swap file should be active and in use. You can verify this by running `free -m` and seeing a swap file listed, with a size greater than 0. While I certainly hope you won't need to resort to using swap, I know from experience that it's only a matter of time. Knowing how to add and activate swap when you need it is definitely a good practice, but for the most part, you should be fine because, by default on most platforms, you'll have swap created for you when setting up Ubuntu for the first time during installation. If you do need to create it manually for whatever reason, I always recommend a bare minimum of 2 GB on servers, or higher if that better fits your use-case.

How much swap is being used is something you should definitely keep an eye on. When the memory starts to get full, the server will start to utilize the swap file. It's normal for a small portion of swap to be utilized even when the majority of the RAM is free. But if a decent chunk of swap is being used, it should be investigated (perhaps a process is using a larger than normal amount of memory).

You can actually control at which point your server will begin to utilize swap. How frequently a Linux server utilizes swap is referred to as its `swappiness`. By default, the `swappiness` value on a Linux server is typically set to `60`. You can verify this with the following command:

```
cat /proc/sys/vm/swappiness
```

The higher the `swappiness` value, the more likely your server will utilize swap. If the `swappiness` value is set to `100`, your server will use swap more often. If you set it to `0`, swap will be used a lot less often. This value correlates roughly to the percentage of RAM being used.

To change this value on the fly, you can execute the following command:

```
sudo sysctl vm.swappiness=30
```

Once you execute that command, the change in the `swappiness` value will take effect immediately. However, once you reboot your server, the value will reset. In order to make the change persist, open the following file with your text editor of choice:

```
/etc/sysctl.conf
```

A line in that file corresponding to `swappiness` will typically not be included by default, but you can add it manually. To do so, add a line such as the following to the end of the file and save it:

```
vm.swappiness = 30
```

Changing this value is one of many techniques within the realm of performance tuning. While the default value of `60` is probably fine for most, there may be a situation where you're running a performance-minded application and can't afford to have it swap any more than it absolutely has to. In such a situation, you would try different values for `swappiness` and use whichever one works best during your performance tests.

In the next section, we'll take a look at another important metric to keep an eye on: load average. The load average gives us an idea of how busy the CPU(s) might be, so we can better understand how to tell when our server is overwhelmed and we may need to take action.

Understanding load average

Another very important topic to understand when monitoring performance is **load average**, which is a series of numbers that represents your server's trend in CPU utilization over a given time. You've probably already seen these series of numbers before, as there are several places in which the load average appears. If you run the `htop` utility, for example, the load average is shown on the screen. In addition, if you execute the `uptime` command, you can see the load average in the output of that command as well. You can also view your load average by viewing the text file that stores it in the first place:

```
cat /proc/loadavg
```

Personally, I habitually use the `uptime` command in order to view the load average. The primary purpose of the `uptime` command is to display the amount of time that the server has been in use, and this time resets anytime you power off or reboot the server. But in addition to how long the server has been powered on, the `uptime` command displays the current load average of your server as well.

Understanding load average might be a bit confusing at first, but you'll quickly realize it's not as complicated as it seems. The load average is a set of three numbers, each corresponding to a portion of time. From left to right, these numbers correspond to 1 minute, 5 minutes, and 15 minutes respectively. A typical load average may look something like the following:

```
0.36, 0.29, 0.31
```

In this example, we have a load average of `0.36` in the 1-minute section, `0.29` in the 5-minute section, and `0.31` in the 15-minute section. In particular, each number represents how many tasks were waiting for attention from the CPU for that given time period. Therefore, these numbers are really good. The server isn't that busy, since virtually no task is waiting for the CPU at any one moment (each number is less than 1). This is contrary to something such as overall CPU percentages, which you may have seen in task managers on other platforms or even within such Linux tools as `htop`. While viewing your CPU usage percentage can be useful, the problem with this is that your CPUs will constantly go back and forth from a high percentage of usage to a low percentage of usage, which you can see for yourself by just running `htop` for a while. When a task does some sort of processing, you might see your cores shoot up to 100 percent and then right back down to a lower number. That really doesn't tell you much, though. With load averages, you're seeing the trend of usage over three given time frames, which is more accurate in determining whether your server's CPUs are running efficiently or are choking on a workload it just can't handle.

The main question, though, is at which point you should start to worry, which really depends on what kind of CPUs are installed on your server. Your server will have one or more CPUs, each with one or more cores. To Linux, each of these cores, whether they are physical or virtual, is the same thing (a CPU). In my case, the machine I took the earlier output from has a CPU with four cores.

The more CPUs your server has, the more tasks it's able to handle at any given time, which also means it can handle a higher load average.

When a load average for a particular time period is equal to the number of CPUs on the system, that means your server is at 100% capacity. It's handling a consistent number of tasks that are equal to the number of tasks it can handle. For example, if you have an 8-core CPU and the load average is 8 for a given time frame, then the CPU is 100% at its available capacity for that time frame. If your load average is consistently more than the number of cores you have available, that's when you'd probably want to look into the situation. It's fine for your server to be at capacity every now and then, but if it always is, that's a cause for alarm.

I'd hate to use a cliché example in order to fully illustrate this concept, but I can't resist, so here goes. A load average on a Linux server is equivalent to the check-out area at a supermarket. A supermarket will have several registers open, where customers can pay to finalize their purchases and move along. In my experience at typical stores in my area, you would have something like 20 check-out registers but only two cashiers working at any one time, but for this example, we'll assume each register has a cashier operating it.

Each cashier is only able to handle one customer at a time. If there are more customers waiting to check out than there are cashiers, the lines will start to back up and customers will get frustrated. In a situation where there are four cashiers and a total of four customers at a particular time, the cashiers would be at capacity, which is not really a big deal since no one else is waiting. What can add to this problem is a customer that is paying by check and/or using a few dozen coupons, which makes the checkout process much longer (similar to a resource-intensive process). If there were four cashiers and six customers waiting, then there would be two more customers than the store is able to handle at the same time. In that case, the checkout area of the store would be above capacity. This is essentially how load average works. Each cashier is a CPU, and each customer is a process that needs CPU time.

Just like the cashiers, each CPU can only handle one task at a time, with some tasks hogging the CPU longer than others. If there are exactly as many tasks as there are CPUs, there's no cause for concern. But if the lines start to back up, we may want to investigate what is taking so long. To gain control of the situation, we may hire an additional cashier (add a new CPU) or ask a disgruntled customer to leave (kill a process).

Let's take a look at another example load average:

```
1.87, 1.53, 1.22
```

In this situation, we shouldn't be concerned, because our hypothetical server has 4 CPUs, and none of them have been at capacity within the 1-, 5-, or 15-minute time periods. Even though the load is consistently higher than 1, we have CPU resources to spare, so it's no big deal. If we had one of those awesome new Threadripper CPUs from AMD (which can contain an impressive number of cores) then those numbers would represent *extremely* low load. Going back to our supermarket comparison, the load average in the previous example would be equivalent to having four cashiers with an average of almost two customers being assisted during any 1 minute. If this server only had one CPU, we would probably want to figure out what's causing the line to begin to back up.

While you might logically assume that having a low load average is a good thing, it can actually represent a really big problem depending on the context. When we deploy servers, we do so to get some sort of work done.

Whether that “work” is to host an application or run jobs to process data, our servers need to be doing some sort of work, otherwise, we’re wasting money by having them. If the load average of your server drops to an abnormally low value, that might mean that a service that would normally be running all the time has failed and exited. For example, if you have a database server that constantly has a load within the 1.x range that suddenly drops to 0.x, that might mean that you either have legitimately less traffic or the database server service is no longer running. This is why it’s always a good idea to develop baselines for your server, in order to gauge what is normal and what isn’t. A baseline refers to resource usage, most of the time. If the resource usage is drastically higher or even lower than the baseline, that’s a potential cause for concern either way.

Overall, load averages are something you’ll become very familiar with as a Linux administrator if you haven’t already. As a snapshot in time of how heavily utilized your server is, it will help you to understand when your server is running efficiently and when it’s having trouble. If a server is having trouble keeping up with the workload you’ve given it, it may be time to consider increasing the number of cores (if you can) or scaling out the workload to additional servers. When troubleshooting utilization, planning for upgrades, or designing a cluster, the process always starts with understanding your server’s load average so you can plan your infrastructure to run efficiently for its designated purpose.

Now that we’ve gone over the important resources that we need to monitor to ensure our server remains healthy, let’s take a look at a useful utility we can utilize that will make resource usage even easier to understand.

Viewing resource usage with htop

When wanting to view the overall performance of your server, nothing beats htop. Although not typically installed by default, htop is one of those utilities that I recommend everyone installs as soon as possible, since it’s indispensable when wanting to check on the resource utilization of your server. It’s so useful in fact that I’ve mentioned it several times earlier in this chapter, even before we started an actual discussion about it in this section. It’s a great utility.

If you don’t already have htop installed, all you need to do is install it with apt:

```
sudo apt install htop
```

When you run htop at your shell prompt, you will see the htop application in all its glory. In some cases, it may be beneficial to run htop as root, since doing so does give you additional options such as being able to kill processes, though this is not required:

```

jay@ubuntu-server: ~
0[          0.0%] Tasks: 27, 30 thr; 1 running
1[          0.0%] Load average: 0.00 0.00 0.00
Mem[|||||||] 183M/1.93G Uptime: 00:40:24
Swp[          0K/2.00G]

  PID USER   PRI  NI  VIRT   RES   SHR  S  CPU% MEM%   TIME+  Command
    1 root     20    0   98M  11432  8136  S   0.0  0.6   0:01.14 /sbin/init
   392 root     19   -1 56016 23576 22460  S   0.0  1.2   0:00.15 /lib/systemd/systemd-journald
   431 root     RT    0  346M 27628  9076  S   0.0  1.4   0:00.28 /sbin/multipathd -d -s
   433 root     20    0  346M 27628  9076  S   0.0  1.4   0:00.00 /sbin/multipathd -d -s
   434 root     RT    0  346M 27628  9076  S   0.0  1.4   0:00.00 /sbin/multipathd -d -s
   435 root     RT    0  346M 27628  9076  S   0.0  1.4   0:00.00 /sbin/multipathd -d -s
   436 root     RT    0  346M 27628  9076  S   0.0  1.4   0:00.01 /sbin/multipathd -d -s
   437 root     RT    0  346M 27628  9076  S   0.0  1.4   0:00.10 /sbin/multipathd -d -s
   438 root     RT    0  346M 27628  9076  S   0.0  1.4   0:00.00 /sbin/multipathd -d -s
   440 root     20    0 25984  6852  4644  S   0.0  0.3   0:00.16 /lib/systemd/systemd-udev
   669 systemd-t 20    0 89300  6464  5660  S   0.0  0.3   0:00.05 /lib/systemd/systemd-timesyncd
   682 systemd-t 20    0 89300  6464  5660  S   0.0  0.3   0:00.00 /lib/systemd/systemd-timesyncd
F1Help F2Setup F3Search F4Filter F5Tree F6SortBy F7Nice -F8Nice +F9Kill F10Quit

```

Figure 8.7: Running `htop`

At the top of the `htop` display, you'll see a progress meter for each of your cores (the server used for my screenshot only has one core), as well as a meter for memory as well as swap. In addition, the upper portion will also show your Uptime, Load average, and the number of Tasks you have running. The lower section of `htop`'s display will show you a list of processes running on your server, with fields showing you useful information such as how much memory or CPU is being consumed by each process, as well as the command being run, the user running it, and its **Process ID (PID)**. We discussed PIDs in *Chapter 7, Controlling and Managing Processes*. To scroll through the list of processes, you can press *Page Up* or *Page Down* or use your arrow keys. In addition, `htop` features mouse support, so you are also able to click on columns at the top in order to sort the list of processes by that criteria. For example, if you click on **MEM%** or **CPU%**, the process list will be sorted by memory or CPU usage respectively. The contents of the display will be updated every 2 seconds.

The `htop` utility is also customizable. If you prefer a different color scheme, for example, you can press *F2* to enter **Setup mode**, navigate to **Colors** on the left, and then you can switch your color scheme to one of the six that are provided. Other options include the ability to add additional meters, add or remove columns, and more. One tweak I find especially helpful on multicore servers is the ability to add an average CPU bar. Normally, `htop` shows you a meter for each core on your server, but if you have more than one, you may be interested in the average as well. To do so, enter **Setup mode** again (*F2*), then with **Meters** highlighted, arrow to the right to highlight **CPU average**, and then press *F5* to add it to the left column. There are other meters you can add as well, such as **Load average**, **Battery**, and more.

Depending on your environment, function keys may not work correctly in terminal programs such as `htop`, because those keys may be mapped to something else. For example, `F10` is used to exit `htop`, but that may not work if `F10` is mapped to a function within your terminal emulator, and using a virtual machine solution such as VirtualBox may also prevent some of these keys from working normally. You can also navigate `htop` with your mouse, even via an SSH connection. This also means that you can click on the word **Quit** in the lower right-hand corner to exit the application.

Here's an example of `htop` configured with a meter for the CPU average:

```

jay@ubuntu-server: ~
0[ | 0.7%] Tasks: 27, 30 thr; 1 running
1[ | 0.0%] Load average: 0.00 0.00 0.00
Avg[ | 0.0%] Uptime: 00:42:18
Mem[ ||||| ] 183M/1.93G
Swp[ | ] 0K/2.00G

PID USER PRI NI VIRT RES SHR S CPU% MEM% TIME+ Command
1 root 20 0 98M 11432 8136 S 0.0 0.6 0:01.14 /sbin/init
392 root 19 -1 56016 23576 22460 S 0.0 1.2 0:00.15 /lib/systemd/systemd-journald
431 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.30 /sbin/multipathd -d -s
433 root 20 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
434 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
435 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
436 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.01 /sbin/multipathd -d -s
437 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.10 /sbin/multipathd -d -s
438 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
440 root 20 0 25984 6852 4644 S 0.0 0.3 0:00.16 /lib/systemd/systemd-udev
669 systemd-t 20 0 89300 6464 5660 S 0.0 0.3 0:00.05 /lib/systemd/systemd-timesyncd
F1Help F2Setup F3Search F4Filter F5Tree F6SortBy F7Nice - F8Nice + F9Kill F10Quit

```

Figure 8.8: `htop` with a meter for CPU average added

When you open `htop`, you will see a list of processes for every user on the system. When you have a situation where you don't already know which user/process is causing extreme load, this is ideal. However, a very useful trick (if you want to watch a specific user) is to press `u` on your keyboard, which will open up the **Show processes of:** menu. In this menu, you can highlight a specific user by highlighting it with the up or down arrow keys and then pressing `Enter` to only show processes for that user. This will greatly narrow down the list of processes.

Another useful view is the **tree view**, which allows you to see a list of processes organized by their parent/child relationship, rather than just a flat list. In practice, it's common for a process to be spawned by another process. In fact, all processes in Linux are spawned from at least one other process, and this view shows that relationship directly. In a situation where you are stopping a process only to have it immediately re-spawn, you would need to know what the parent of that process is in order to stop it from resurrecting itself. Pressing `F5` will switch `htop` to tree view mode, and pressing it again will disable the tree view.

With the tree view activated, htop will appear similar to the following:

```

jay@ubuntu-server: ~
0.0%] Tasks: 27, 30 thr; 1 running
1|] Load average: 0.00 0.00 0.00
Avg[|] 0.7%] Uptime: 00:57:14
Mem[|] 178M/1.93G
Swp[|] 0K/2.00G

PID/USER PRI NI VIRT RES SHR S CPU% MEM% TIME+ Command
1 root 20 0 98M 11436 8136 S 0.0 0.6 0:01.17 /sbin/init
392 root 19 -1 56016 23724 22608 S 0.0 1.2 0:00.16 /lib/systemd/systemd-journald
431 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.37 /sbin/multipathd -d -s
433 root 20 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
434 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
435 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
436 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.01 /sbin/multipathd -d -s
437 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.13 /sbin/multipathd -d -s
438 root RT 0 346M 27628 9076 S 0.0 1.4 0:00.00 /sbin/multipathd -d -s
440 root 20 0 25984 6852 4644 S 0.0 0.3 0:00.16 /lib/systemd/systemd-udev
669 systemd-t 20 0 89300 6464 5660 S 0.0 0.3 0:00.05 /lib/systemd/systemd-timesyncd
682 systemd-t 20 0 89300 6464 5660 S 0.0 0.3 0:00.00 /lib/systemd/systemd-timesyncd
710 systemd-n 20 0 16068 7956 6944 S 0.0 0.4 0:00.03 /lib/systemd/systemd-networkd
712 systemd-r 20 0 25220 12300 8276 S 0.0 0.6 0:00.06 /lib/systemd/systemd-resolved
723 root 20 0 6892 2856 2612 S 0.0 0.1 0:00.00 /usr/sbin/cron -f -P
724 messagebu 20 0 8768 4860 4032 S 0.0 0.2 0:00.06 @dbus-daemon --system --address=systemd:

F1 Help F2 Setup F3 Search F4 Filter F5 List F6 SortBy F7 Nice F8 vice F9 Kill F10 Quit

```

Figure 8.9: htop with tree view activated

As I've mentioned, htop updates its stats every 2 seconds by default. Personally, I find this to be acceptable, but if you want to change how fast it refreshes, you can call htop with the `-d` option and then apply a different number of seconds (entered in tenths of seconds) for it to refresh. For example, to run htop but have it update every 7 seconds, start htop with the following command:

```
htop -d 70
```

To kill a process with htop, use your up and down arrow keys to highlight the process you wish to kill and press `F9`. A new menu will appear, giving you a list of signals you are able to send to the process with htop. `SIGTERM`, as we discussed before, will attempt to gracefully terminate the process. `SIGKILL` will terminate it uncleanly. Once you highlight the signal you wish to send, you can send it by pressing `Enter` or cancel the process with `Esc`.

As you can see, htop can be incredibly useful and has (for the most part) replaced the legacy `top` command that was popular in the past for most administrators. The `top` command is available by default in Ubuntu Server and is worth a look, if only as a comparison to htop. Like htop, the `top` command gives you a list of processes running on your server, as well as their resource usage. There are no pretty meters and there is less customization possible, but the `top` command serves the same purpose. In most cases, though, htop is probably your best bet going forward.

Summary

In this chapter, we learned how to monitor our server's resource usage. We began with a look at the commands we can use to investigate disk usage, and we learned how to monitor memory usage as well. We also discussed swap, including what it is, why you'd want to have it, as well as how to create a swap file manually should the need to do so come up. We then took a look at load average and closed out the chapter by checking out `htop`, which is my favorite utility for getting an overall look at resource usage on servers.

In *Chapter 9, Managing Storage Volumes*, we'll take a closer look at storage. In this chapter, we learned how to see how much is being used, but in the next, we'll look at more advanced concepts surrounding storage, such as formatting volumes, adding additional volumes, and even LVM. See you there!

Relevant videos

- Linux Crash Course – The `du` Command (LearnLinuxTV): <https://linux.video/du>
- Linux Crash Course – `htop` (LearnLinuxTV): <https://linux.video/htop>
- Linux Crash Course – Load Average (LearnLinuxTV): <https://linux.video/loadavg>
- Linux Crash Course – Understanding Memory Usage (LearnLinuxTV): <https://linux.video/mem>
- Awesome Linux Tools - `ncdu` (LearnLinuxTV): <https://linux.video/ncdu>

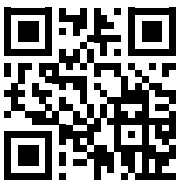
Further reading

- Linux ate my RAM: <https://learnlinux.link/ate-ram>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



9

Managing Storage Volumes

When it comes to storage on our servers, it seems as though we can never get enough. While hard disks are growing in capacity every year, and high-capacity disks are cheaper than ever, our servers gobble up available space quickly. As administrators of servers, we always do our best to order servers with ample storage, but business needs evolve over time, and no matter how well we plan, a successful business will always need more. While managing your servers, you'll likely find yourself adding additional storage at some point. But managing storage is more than just adding new disks every time your current one gets full. Planning ahead is also important, and technologies such as **Logical Volume Manager (LVM)** will make your job much easier as long as you start using it as early as you possibly can.

LVM itself is just one of the concepts we'll go over in this chapter that will give you more flexibility with how you handle servers. I'll also walk you through additional concepts that will no doubt come in handy as you manage storage and volumes on your server. More specifically, this discussion will include:

- Adding additional storage volumes to the filesystem
- Formatting and partitioning storage devices
- Mounting and unmounting volumes
- Understanding the `/etc/fstab` file
- Backing up and restoring volumes
- Utilizing LVM

One possible solution when your server is running out of disk space is to add an additional storage volume. So the first order of business for us, in this chapter, will be to look into how we can do exactly that.

Adding additional storage volumes

At some point or another, you'll reach a situation where you'll need to add additional storage to your server. On physical servers, we can add additional hard disks, and on virtual or cloud servers, we can add additional virtual disks. Either way, in order to take advantage of the extra storage, we'll need to determine the name of the device, format it, and mount it.

In the case of LVM (which we'll discuss later in this chapter), we'll have the opportunity to expand an existing volume, often without a server reboot being necessary. There's an overall process to follow when adding a new device, though. When adding additional storage to your system, you should ask yourself the following questions:

How much storage do you need? If you're adding a virtual disk, you can usually make it any size you want, as long as you have enough space remaining in the pool of your hypervisor.

After you attached it, what device name did it receive? When a new disk is attached to our server, it will be detected by the system and given a device name. In most cases, the naming convention of `/dev/sda`, `/dev/sdb`, and so on will be used. In other cases (such as virtual disks), this will be different, such as `/dev/vda`, `/dev/xda`, and possibly others. The naming scheme usually ends with a letter, incrementing to the next letter with each additional disk.

How do you want the storage device formatted? At the time of writing, the ext4 filesystem is the most common. However, for different workloads, you may consider other options (such as XFS). When in doubt, use ext4, but definitely read up on the other options to see if they may benefit your use case. ZFS is another option that you can consider, though compared to the other choices, it's relatively new. We'll discuss formatting in the next section, *Formatting and partitioning storage devices*.

It may be common knowledge to you by now, but the word filesystem is a term that can have multiple meanings on a Linux system depending on its context and may confuse newcomers. Linux administrators like us will most often use the term filesystem to discuss the file and directory structure of a typical Linux system. However, the term is also used to describe how a disk is formatted for use with the distribution (for example, the ext4 filesystem). In this chapter, we'll be primarily focusing on the latter.

Where do you want it mounted? The new disk needs to be accessible to the system and possibly users, so you would want to mount (attach) it to a directory on your filesystem where your users or your application will be able to use it. In the case of LVM, which we also discuss in this chapter, you're probably going to want to attach it to an existing storage group. You can come up with your own directory for use with the new volume, but I'll discuss a few common locations later on in this chapter. We'll go over the process of mounting and unmounting in the *Mounting and unmounting volumes* section.

Let's consider the answers to the first two questions. With regard to how much space you should add, you would want to research the needs of your application or organization and find a reasonable amount. In the case of physical disks, you don't really get a choice beyond deciding which disk to purchase. In the case of virtual disks, you're able to be more frugal, as you can add a small disk to meet your needs (you can always add more later).

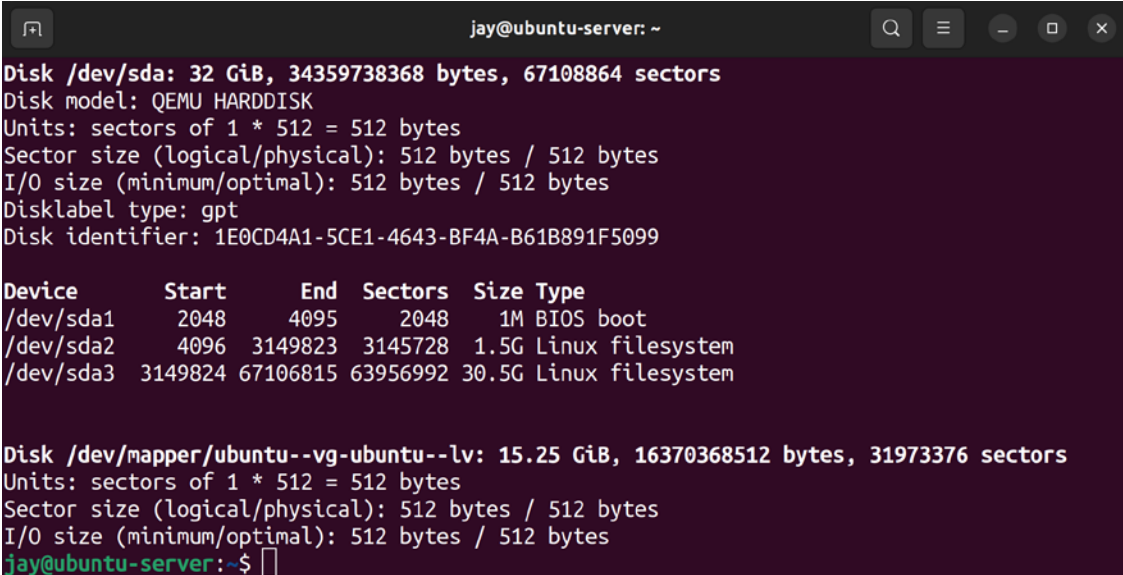
The main benefit of LVM with virtual disks is being able to grow a filesystem without a server reboot. For example, you can start with a 30 GB volume and then expand it in increments of 10 GB by adding additional 10 GB virtual disks. This method is certainly better than adding a 200 GB volume all at once when you're not completely sure all that space will ever be used.

LVM can also be used on physical servers as well, but would most likely require a reboot anyway since you'd have to open the case and physically attach a hard drive. Some servers allow for hot-plugging, which gives you the ability to add/remove physical hard drives without powering off the server first, which is a great benefit to have.

Next, the device name can be found with the `fdisk -l` command. The `fdisk` command is normally used for creating and deleting partitions, but it will also allow us to determine which device name our new disk received. Using the `fdisk -l` command will give you the info, but you'll need to run it as root or with `sudo`:

```
sudo fdisk -l
```

Executing this command produces output similar to the following:



```
jay@ubuntu-server: ~  
Disk /dev/sda: 32 GiB, 34359738368 bytes, 67108864 sectors  
Disk model: QEMU HARDDISK  
Units: sectors of 1 * 512 = 512 bytes  
Sector size (logical/physical): 512 bytes / 512 bytes  
I/O size (minimum/optimal): 512 bytes / 512 bytes  
Disklabel type: gpt  
Disk identifier: 1E0CD4A1-5CE1-4643-BF4A-B61B891F5099  
  
Device          Start      End  Sectors  Size Type  
/dev/sda1       2048      4095    2048    1M BIOS boot  
/dev/sda2       4096  3149823  3145728  1.5G Linux filesystem  
/dev/sda3     3149824  67106815  63956992  30.5G Linux filesystem  
  
Disk /dev/mapper/ubuntu--vg-ubuntu--lv: 15.25 GiB, 16370368512 bytes, 31973376 sectors  
Units: sectors of 1 * 512 = 512 bytes  
Sector size (logical/physical): 512 bytes / 512 bytes  
I/O size (minimum/optimal): 512 bytes / 512 bytes  
jay@ubuntu-server:~$
```

Figure 9.1: Output of the `fdisk -l` command

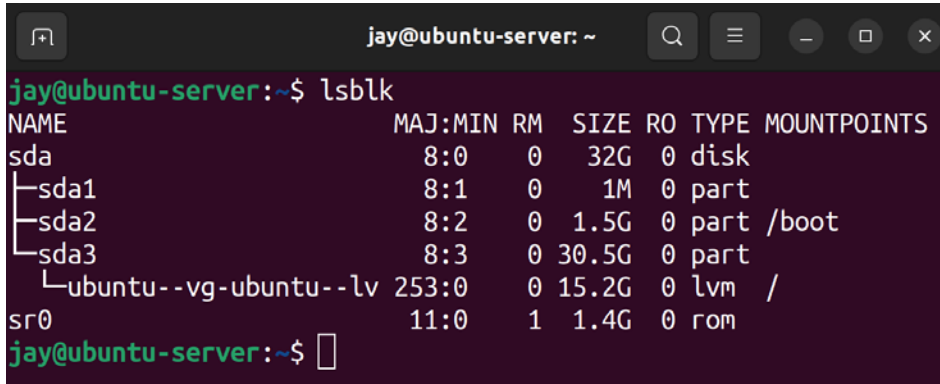
I always recommend running `fdisk -l` *before and after* attaching a new device. That way, it will be more obvious which device name represents the new device.

Another trick is to use the following command, with which the output will update automatically as you add the disk:

```
dmesg --follow
```

Just start the command, attach the disk, and watch the output. When done, press `Ctrl + c` on your keyboard to return to the prompt.

You can also find the device name of your new disk with the `lsblk` command. The benefit of `lsblk` is that you don't need root privileges, and the information it returns is simplified:



```

jay@ubuntu-server: ~
┌───┴───
jay@ubuntu-server:~$ lsblk
NAME                                MAJ:MIN RM  SIZE RO TYPE MOUNTPOINTS
sda                                  8:0    0   32G  0 disk
├──sda1                              8:1    0    1M  0 part
├──sda2                              8:2    0   1.5G  0 part /boot
├──sda3                              8:3    0  30.5G  0 part
└──ubuntu--vg-ubuntu--lv 253:0    0  15.2G  0 lvm /
sr0                                  11:0    1   1.4G  0 rom
jay@ubuntu-server:~$

```

Figure 9.2: Output of the `lsblk` command

On a typical server, the first disk (basically, the one that you installed Ubuntu Server on) will be given a device name of `/dev/sda` while additional disks will be given the next available name, such as `/dev/sdb`, `/dev/sdc`, and so on (depending on the type of hard disk you have). Nowadays, **Non-Volatile Memory Express (NVMe)** hard drives are becoming increasingly common, so you may see a device name similar to `/dev/nvme0n1`. You'll also need to know the partition number. Device names for disks will also have numbers at the end, representing individual partitions. For example, the first partition of `/dev/sda` will be given `/dev/sda1`, while the second partition of `/dev/sdc` will be given `/dev/sdc2`. These numbers increment and are often easy to predict. As I mentioned before, your device naming convention may vary from server to server, especially if you're using a **Redundant Array of Independent Disks (RAID)** controller or a virtualization host such as VMware or XenServer. If you haven't created a partition on your new disk yet, you will not see any partition numbers at the end of their names.

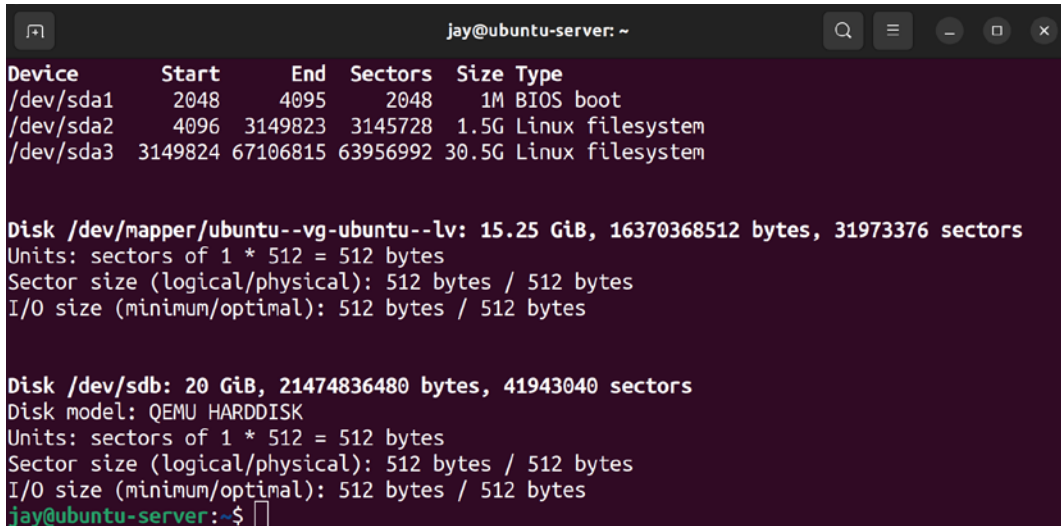
Now that you've added and named an additional storage volume, we can proceed through the process of setting it up. We need to decide where we're going to mount it, and what purpose it will serve. But before we can even mount the storage device in the first place, we need to create at least one partition on it and then format it. We'll take care of both in the next section.

Formatting and partitioning storage devices

Once you've installed a physical or virtual disk, you're well on your way to benefiting from additional storage. But in order to utilize a disk, it must first be formatted. In order to ensure we're formatting the correct disk, we need to find the name the device was given. As you already know from the previous section, there's a specific naming scheme that is used in Linux distributions to name disks. So you should already know the device name of the new disk. As explained earlier, you can use the `sudo fdisk -l` command to see details regarding the storage devices attached to your server:

```
sudo fdisk -l
```

This will produce an output that looks similar to the following:



```
jay@ubuntu-server: ~  
Device      Start      End  Sectors  Size Type  
/dev/sda1   2048      4095    2048    1M BIOS boot  
/dev/sda2   4096   3149823  3145728  1.5G Linux filesystem  
/dev/sda3  3149824  67106815  63956992 30.5G Linux filesystem  
  
Disk /dev/mapper/ubuntu--vg-ubuntu--lv: 15.25 GiB, 16370368512 bytes, 31973376 sectors  
Units: sectors of 1 * 512 = 512 bytes  
Sector size (logical/physical): 512 bytes / 512 bytes  
I/O size (minimum/optimal): 512 bytes / 512 bytes  
  
Disk /dev/sdb: 20 GiB, 21474836480 bytes, 41943040 sectors  
Disk model: QEMU HARDDISK  
Units: sectors of 1 * 512 = 512 bytes  
Sector size (logical/physical): 512 bytes / 512 bytes  
I/O size (minimum/optimal): 512 bytes / 512 bytes  
jay@ubuntu-server: ~$
```

Figure 9.3: Using `fdisk -l` to view a list of storage devices on the server

In my case, the device `/dev/sdb` is brand-new—I just added it to the server. Since I’m using a virtual machine for the examples in this chapter, the new disk shows a model of `QEMU HARDDISK`. It doesn’t have any partitions currently; notice how we see a few lines above it referring to a different hard disk and partitions, such as `/dev/sda3`. We don’t have any lines like that in the description underneath for `/dev/sdb`. If we did have one or more partitions on that device, they would show up in the output.


At this point, we know which storage device is new—there’s no doubt that in the preceding example, it’s `/dev/sdb`. We always need to make sure we don’t attempt to format or repartition the wrong device, or we may lose data. In this case, we can see `/dev/sdb` has no partitions (and this volume wasn’t present before I added it), so it’s obvious which disk we’ll want to work with. Now we can create one or more partitions on it, to continue preparing it for use.

Creating a partition

To create an actual partition on this device, we’ll use the `fdisk` command with `sudo`, using the device’s name as an option. In my case, I would execute the following to work with disk `/dev/sdb`:

```
sudo fdisk /dev/sdb
```

Note that I didn't include a partition number here, as `fdisk` works with the disk directly (and we also have yet to create any partitions). In this section, I'm assuming you have a disk that has yet to be partitioned or one you won't mind wiping. When executed correctly, `fdisk` will show you an introductory message and give you a prompt:

A terminal window screenshot showing the execution of the `fdisk` utility. The prompt is `jay@ubuntu-server:~$ sudo fdisk /dev/sdb`. The output shows the welcome message, a warning about memory changes, and the creation of a new DOS disklabel with identifier `0xe8bfef0`. The prompt `Command (m for help):` is shown with a cursor in a box.

```
jay@ubuntu-server:~$ sudo fdisk /dev/sdb
Welcome to fdisk (util-linux 2.37.2).
Changes will remain in memory only, until you decide to write them.
Be careful before using the write command.

Device does not contain a recognized partition table.
Created a new DOS disklabel with disk identifier 0xe8bfef0.

Command (m for help):
```

Figure 9.4: Main prompt of `fdisk`

At this point, you can press `m` on your keyboard for a menu of possible commands you can execute. In this example, I'll walk you through the commands required to set up a new disk for the first time.

I'm sure it goes without saying, but be aware of the destructive possibilities of `fdisk`. If you run `fdisk` against the wrong drive, irrecoverable data loss may result. It's common for an administrator to memorize utilities such as `fdisk` to the point where using it becomes muscle memory. But always take the time to ensure that you're running such commands against the appropriate disk.

Before we continue with creating a new partition, some discussion is required with regard to the **Master Boot Record (MBR)** and **GUID Partition Table (GPT)** partition tables. When creating a partition table on a new disk, you'll have the option to set it up to use an MBR or GPT partition table. GPT is the newer standard, while MBR has been around for quite some time and is probably what you've been using if you've been working with servers for a long time.

You may see MBR referred to as DOS when referring to the older partition structure. As you may already know, **DOS** is short for **Disk Operating System**, but we're not referencing that operating system here in this chapter; we're referencing the partitioning structure that IBM came up with decades ago. For example, while using `fdisk`, it will refer to the MBR partition structure as DOS. In this chapter, we'll use MBR to refer to the older style whenever possible to avoid confusion.

With MBR partition tables, you have some limitations to consider. First, MBR only allows you to create up to four primary partitions. In addition, it also limits you to using somewhere around 2 TB of a disk. If the capacity of your disk is 2 TB or less, this won't be an issue. However, disks larger than 2 TB are becoming more and more common.

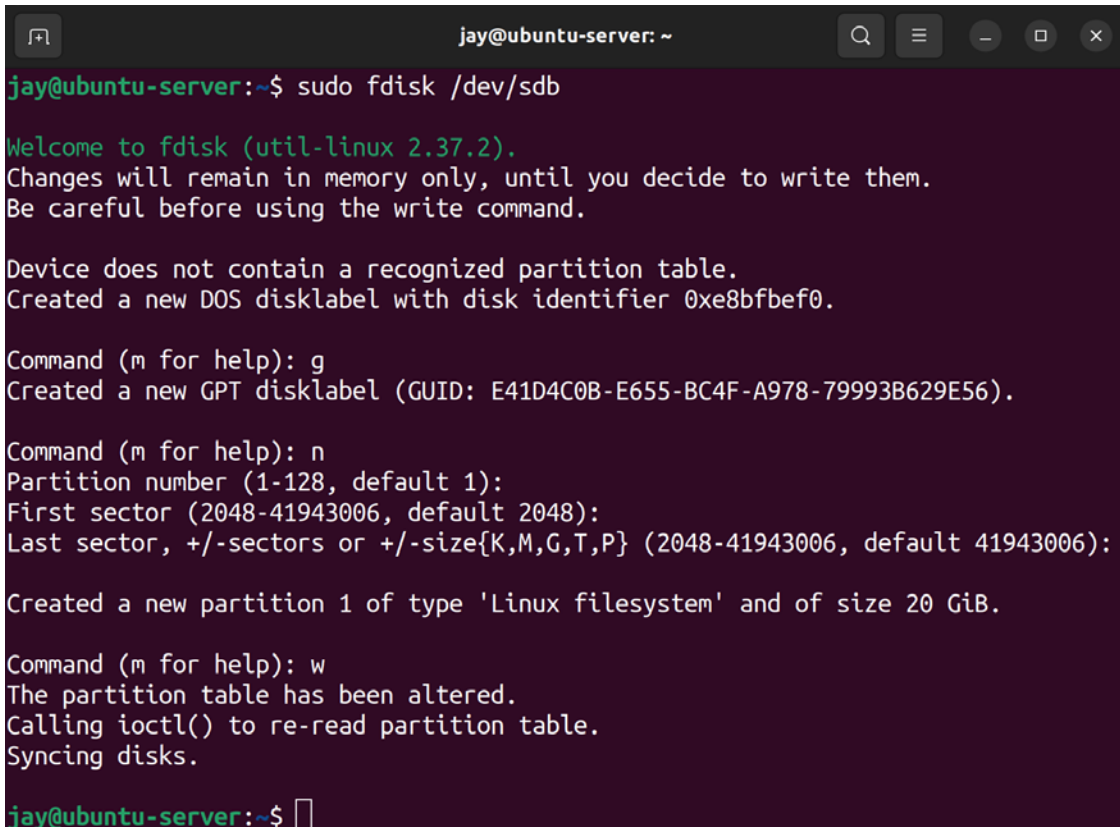
On the other hand, GPT doesn't have a 2 TB restriction, so if you have a very large disk, the decision between MBR and GPT has pretty much been made for you. In addition, GPT doesn't have a restriction of up to four primary partitions, as `fdisk` with a GPT partition table will allow you to create up to 128 of them. It's certainly no wonder GPT is fast becoming the new standard! It's only a matter of time before GPT becomes the default, so unless you have a good reason not to, I recommend using it if you have a choice.

When you first enter the `fdisk` prompt, you can press `o` to create an MBR-style partition layout, or you can press `g` to create the partition layout with the newer GPT style. As I've mentioned before, this is a potentially destructive process, so make sure you're using this utility against the correct drive! Make sure you press the associated key for your chosen partition style and then press *Enter*, and then we can proceed. Once you press `g` or `o`, you should see a confirmation that you have created a new partition table.

Continuing on, after you've made your choice and created either an MBR or GPT partition table, we're ready to proceed. Next, at the `fdisk` prompt, type `n` to tell `fdisk` that you would like to create a new partition. Then, you'll be asked if you would like to create a primary or extended partition (if you've opted for MBR). With MBR, you would want to choose primary for the first partition, and then you can use extended for creating additional partitions. If you've opted for GPT, this prompt won't appear, as it will create your partition as primary no matter what.

The next prompt that will come up will ask you for the partition number, defaulting to the next available number. Press *Enter* to accept the default. Afterward, you'll be asked for the first sector of the partition to use (press *Enter* to accept the default of 2,048), and then the next prompt will ask you for the last sector to use. If you press *Enter* to accept the default last sector, your partition will consist of all the free space that was remaining on the device. If you'd like to create multiple partitions, don't accept the default at the last sector prompt. Instead, you can clarify the size of your new partition by typing the `+` symbol followed by the number of mebibytes or gibibytes to use, and then `M` for mebibytes or `G` for gibibytes. For example, you can enter `+20G` here to create a partition of 20 GiB. Note that there is no space after the `+` symbol, nor is there a space between `20` and `G`.

At this point, you'll be returned to the `fdisk` prompt. To save your changes and exit `fdisk`, press `w` and then `Enter`. Now if you run the `fdisk -l` command as root, you should see the new partition you created. Here is some example output from the `fdisk` command from one of my servers, to give you an idea of what the entire process looks like:



```
jay@ubuntu-server: ~  
jay@ubuntu-server:~$ sudo fdisk /dev/sdb  
Welcome to fdisk (util-linux 2.37.2).  
Changes will remain in memory only, until you decide to write them.  
Be careful before using the write command.  
  
Device does not contain a recognized partition table.  
Created a new DOS disklabel with disk identifier 0xe8bfbef0.  
  
Command (m for help): g  
Created a new GPT disklabel (GUID: E41D4C0B-E655-BC4F-A978-79993B629E56).  
  
Command (m for help): n  
Partition number (1-128, default 1):  
First sector (2048-41943006, default 2048):  
Last sector, +/-sectors or +/-size{K,M,G,T,P} (2048-41943006, default 41943006):  
  
Created a new partition 1 of type 'Linux filesystem' and of size 20 GiB.  
  
Command (m for help): w  
The partition table has been altered.  
Calling ioctl() to re-read partition table.  
Syncing disks.  
  
jay@ubuntu-server:~$
```

Figure 9.5: Example run of the `fdisk` command

If you've made a mistake or you want to redo your partition layout, you can do so by entering the `fdisk` prompt again and then pressing `g` to create a new GPT layout or `o` to create a new MBR layout. Then, continue through the steps again to partition your disk. Feel free to practice this a few times until you get the hang of the process.

Formatting partitions

After you create your partition layout for your new disk and you're satisfied with it, you're ready to format it. Now that I've created a partition layout on the new disk, the output of `sudo fdisk -l` will be different:

```

jay@ubuntu-server: ~
Disk /dev/sdb: 20 GiB, 21474836480 bytes, 41943040 sectors
Disk model: QEMU HARDDISK
Units: sectors of 1 * 512 = 512 bytes
Sector size (logical/physical): 512 bytes / 512 bytes
I/O size (minimum/optimal): 512 bytes / 512 bytes
Disklabel type: gpt
Disk identifier: E41D4C0B-E655-BC4F-A978-79993B629E56

Device      Start      End  Sectors  Size Type
/dev/sdb1   2048 41943006 41940959  20G Linux filesystem
jay@ubuntu-server:~$ █

```

Figure 9.6: Another example of `sudo fdisk -l` after creating a partition

Notice that now, we have the partition `/dev/sdb1` added, which is visible in the output. Now, we can go ahead and format it. To do so, we take care of that with the `mkfs` command. This command is run with a specific syntax that entails typing `mkfs` along with a period (`.`), followed by the type of filesystem you would like to format the target as. The following example will format `/dev/sdb1` as `ext4`:

```
sudo mkfs.ext4 /dev/sdb1
```

Your output will look similar to mine in the following screenshot:

```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ sudo mkfs.ext4 /dev/sdb1
mke2fs 1.46.5 (30-Dec-2021)
Discarding device blocks: done
Creating filesystem with 5242619 4k blocks and 1310720 inodes
Filesystem UUID: 117910ed-0fb3-4ee9-9c40-db9473ee7e50
Superblock backups stored on blocks:
    32768, 98304, 163840, 229376, 294912, 819200, 884736, 1605632, 2654208,
    4096000

Allocating group tables: done
Writing inode tables: done
Creating journal (32768 blocks): done
Writing superblocks and filesystem accounting information: done

jay@ubuntu-server:~$ █

```

Figure 9.7: Formatting a volume using the `ext4` filesystem

If you've opted for a filesystem type other than `ext4`, you can use that in place of `ext4` when using `mkfs`. The following example creates an XFS filesystem instead:

```
sudo mkfs.xfs /dev/sdb1
```


Some filesystems, such as XFS, are not supported by default and may need an additional package installed in order for them to be used. In the case of XFS, it requires the `xfsprogs` package to be installed.

So, now that we've created one or more partitions and formatted them, we're ready to mount the newly created partition(s) on our server. In the next section, I'll walk you through mounting and unmounting storage volumes.

Mounting and unmounting volumes

Now that you've added a new storage volume to your server and formatted it, you can mount the new device so that you can start using it. To do this, we use the `mount` command. This command allows you to attach a storage device (or even a network share) to a local directory on your server. Before mounting, the directory must be empty. The `mount` command, which we'll get to practice with an example very shortly, basically just requires you to designate a place (directory) for the device to be mounted to. But where should you mount the volume?

Normally, there are two directories, created by default, in your Ubuntu Server installation that exist for the purposes of mounting volumes: `/mnt` and `/media`. While there is no hard rule as far as where media needs to be mounted, these two directories exist as part of the **Filesystem Hierarchy Standard (FHS)** that was mentioned in *Chapter 4, Navigating and Essential Commands*. The purposes of the `/mnt` and `/media` directories are defined within this specification. The FHS defines `/mnt` as a mount point for a temporarily mounted filesystem, and `/media` as a mount point for removable media.

In plain English, this means that the intended purpose of `/mnt` is for storage volumes you generally keep mounted most of the time, such as additional hard drives, virtual hard disks, and network-attached storage. The FHS document uses the term *temporary* when describing `/mnt`, but in practice, this is typically where things are mounted that you generally expect to be around for a while. In regard to `/media`, the FHS is basically indicating that removable media (flash drives, CD-ROM media, external hard drives, and so on) are intended to be mounted there.

However, it's important to point out that where the FHS indicates you should mount your extra volumes is only a suggestion. (Perhaps a strong suggestion, but a suggestion nonetheless.) No one is going to force you to follow it, and the fate of the world isn't dependent on your choice. With the `mount` command, you can literally mount your extra storage anywhere that isn't already mounted or full of files. You could even create the directory `/kittens` and mount your disks there and you won't suffer any consequences other than a few chuckles from your colleagues.

Often, organizations will come up with their own scheme for where to mount extra disks. Although I personally follow the FHS designation, one example of a custom layout was with a company I worked with in the past. They used the `/store` directory for mounting storage on their servers, a directory they created themselves on each server. Whatever scheme you use is up to you; the only suggestion I can make is to be as consistent as you can from one server to another, if only for the sake of sanity.

The `mount` command generally needs to be run as `root`. While there is a way around that (you can allow normal users to mount volumes, but we won't get into that just yet), it's usually the case that only `root` can or should be mounting volumes. As I mentioned, you'll need a place to mount these volumes, so to facilitate that, we can create a directory called `/mnt/vol1` with the following command:

```
sudo mkdir /mnt/vol1
```

When you've created a directory, like I have, or decided on an existing one, you can mount a volume with a command similar to the following:

```
sudo mount /dev/sdb1 /mnt/vol1
```

In that example, I'm mounting device `/dev/sdb1` to directory `/mnt/vol1`.

Of course, you'll need to adjust the command to reference the device you want to mount and where you want to mount it. As a reminder, if you don't remember which devices exist on your server, you can list them with `fdisk -l`.

Normally, the `mount` command wants you to issue the `-t` option with a given type. In my case, the `mount` command would've been the following had I used the `-t` option, considering my disk is formatted with `ext4`:

```
sudo mount /dev/sdb1 -t ext4 /mnt/vol1
```

A useful trick when mounting devices is to execute the `df -h` command before and after mounting.

While that command is generally used to check how much free space you have on various mounts, it does show you a list of mounted devices, so you can simply compare the results after mounting the device to confirm that it is present.

In that example, I used the `-t` option along with the type of filesystem I formatted the device with. In the first example, I didn't. This is because, in most cases, the `mount` command is able to determine which type of filesystem the device uses and adjust itself accordingly. Thus, most of the time, you won't need the `-t` option. In the past, you almost always needed it, but it's easier nowadays. The reason I bring this up is that if you ever see an error when trying to mount a filesystem that indicates an invalid filesystem type, you may have to specify this. Feel free to check the man pages for the `mount` command for more information regarding the different types of options you can use.

When you are finished using a volume, you can unmount it with the `umount` command (the missing `n` in the word *unmount* is intentional):

```
sudo umount /mnt/vol1
```

The `umount` command, which also needs to be run as `root` or with `sudo`, allows you to disconnect a storage device from your filesystem. In order for this command to be successful, the volume should not be in use. If it is, you may receive a device- or resource-busy error message. If you execute `df -h` after unmounting, you should see that the filesystem is missing from the output and, thus, isn't mounted anymore.

The downside to manually mounting devices is that they will not automatically remount themselves the next time your server boots. In order to ensure the mount is available anytime your server boots up, you'll need to edit the `/etc/fstab` file, which I'll walk you through in the next section.

Understanding the `/etc/fstab` file

The `/etc/fstab` file is a very critical file on your Linux system. You can edit this file to call out additional volumes you would like to automatically mount at boot time. However, the main purpose of this file is to also mount your main filesystem, so if you make a mistake while editing it, your server will not boot (at all). Definitely be careful here.

Analyzing the contents of `/etc/fstab`

When your system boots, it looks at the `/etc/fstab` file to determine where the root filesystem is. In addition, the location of your swap area is read from this file and mounted at boot time as well. Your system will also read any other mount points listed in this file, one per line, and mount them. Basically, just about any kind of storage you can think of can be added to this file and automatically mounted. Even network shares from Windows servers can be added here. It won't judge you (unless you make a typo).

As an example, here are the contents of `/etc/fstab` on one of my machines:



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ cat /etc/fstab
# /etc/fstab: static file system information.
#
# Use 'blkid' to print the universally unique identifier for a
# device; this may be used with UUID= as a more robust way to name devices
# that works even if disks are added and removed. See fstab(5).
#
# <file system> <mount point> <type> <options> <dump> <pass>
# / was on /dev/ubuntu-vg/ubuntu-lv during curtin installation
/dev/disk/by-id/dm-uuid-LVM-dTs7ZjscXf0LLczXWK61mo7EuybQKBbKRmtCaIzd8gt56KLTNJKv0Zr1u7FWAkFV / ext4 defaults 0 1
# /boot was on /dev/sda2 during curtin installation
/dev/disk/by-uuid/30912eb2-fac6-435b-b55b-0d94dafa36a1 /boot ext4 defaults 0 1
/swap.img none swap sw 0 0
jay@ubuntu-server:~$

```

Figure 9.8: Viewing the contents of the `/etc/fstab` file

When you install Ubuntu Server, the `/etc/fstab` file is created for you and populated with a line for each of the partitions the installer created during installation. On the server I used to grab the example `fstab` content, I have a single partition for the root filesystem, and you can also see where the swap file is mentioned.

Each partition is typically designated with a **Universally Unique Identifier (UUID)** instead of the `/dev/sdaX` naming convention you might be more accustomed to if you've worked with storage devices in the past. In my output, you can see UUID `dm-uuid-LVM-H8VEs7qDbMgv...`, which refers to my root filesystem, and you can also see that I have a swap file located at `/swap.img`.

The concept of a UUID has been around for a while, but there's nothing stopping you from replacing the UUID with the actual partition names (such as `/dev/sda1` or similar). If you were to do that, the server would still boot, and you probably wouldn't notice a difference (assuming you didn't make a typo).

Nowadays, UUIDs are preferred over common device names due to the fact that the names of devices can change depending on where you place them physically (such as which particular **Serial Advanced Technology Attachment (SATA)** port a hard disk is plugged into, which of your USB ports an external drive is connected to, and so on) or how you order them (in the case of virtual disks).

Add to this the fact that removable media can be inserted or removed at any time, and you have a situation where you don't really know what name each device is going to have at any one time. For example, your external hard drive may be named `/dev/sdb1` on your system now, but it may not be the next time you mount it if something else you connect claims the name of `/dev/sdb1`. This is one situation in which the concept of UUIDs comes in handy. The UUID of a device will not change if you reorder your disks (but it will change if you reformat the volume). As stated in *Figure 9.8*, you can easily list the UUIDs of your volumes with the `blkid` command:

```
blkid
```

The output will show you the UUID of each device attached to your system, and you can use this command any time you add new volumes to your server to list your UUIDs. This is also the first step in adding a new volume to your `/etc/fstab` file. While I did say that using UUIDs is not required, it's definitely recommended and can save you from trouble later on.

Each line of an `fstab` entry is broken down into several columns, each separated by spaces or tabs. There isn't a set number of spaces necessary to separate each column; in most cases, spaces are only used to line up each column to make them easier to read. However, at least one space is required.

In the first column of the example `fstab` file, we have the device identifier, which can be the UUID or label of each device that differentiates it from the others. (You can add a label to a device while formatting it with the `-L` argument with `mkfs` commands.) In the second column, we have the location we want the device to be mounted to. In the case of the root filesystem, this is `/`, which (as you know) is the beginning of the Linux filesystem. The third entry in the screenshot (for `swap`) has a mount point of `none`, which means that a mount point is not necessary for this device. In the third column, we have the filesystem type, the first two being `ext4`, and the third having a type of `swap`.

In the fourth column, we have a list of options for each mount separated by a comma. In this case, we only have one option for each of the example lines. With the root filesystem, we have an option of `errors=remount-ro`, which tells the system to remount the filesystem as read-only if an error occurs. Such an issue is rare but will keep your system running in read-only mode as best it can if something goes wrong. The `swap` partition has a single option of `sw`. There are many other options that can be used here, so feel free to consult the man pages for a list. We will go over some of these options in this section.

The fifth and sixth columns refer to `dump` and `pass` respectively, which on my system are `0` and `0` for each line. The `dump` partition is almost always `0` and can be used with a backup utility to determine whether the filesystem should be backed up (`0` for no, and `1` for yes). In most cases, just leave this at `0` since this is rarely ever used by anything nowadays. The `pass` field refers to the order in which `fsck` will check the filesystems. The `fsck` utility scans hard disks for filesystem errors in the case of a system failure or a scheduled scan. The possible options for `pass` are `0`, `1`, or `2`. With `0`, the partition is never checked with `fsck`. If set to `1`, the partition is checked first. Partitions with a `pass` of `2` are considered a second priority and checked last. As a general rule of thumb, consider using `1` for your main filesystem and `2` for all others. It's not uncommon for cloud server providers to use `0` for both fields. This may be because if a disk does undergo a routine check, it would take considerably longer to boot up. In a cloud environment, you can't always wait very long to get a server up and running.

Now that we understand all the columns of a typical `fstab` entry, we can work through adding another volume to the `fstab` file.

Adding to the `/etc/fstab` file

To add another volume to the `fstab` file, we first need to know the UUID of the volume we would like to add (assuming it's a hard disk or virtual disk). Again, we do that with the `blkid` command:

```
blkid /dev/sdb1
```

Notice that I used the device name of `/dev/sdb1` as an argument. This is because I want to specifically fetch the UUID of the new device we added. The output of that command will give us the UUID of that device, so it can be added to the `/etc/fstab` file. Copy that down somewhere, as we'll need it shortly. Next, we need to know where we want to mount the volume. Go ahead and create the directory now, or use an existing directory if you wish. For example, you could create the directory `/mnt/extra_storage` for this purpose:

```
sudo mkdir /mnt/extra_storage
```

At this point, we should have all we need in order to add a new entry to `fstab`. To do so, we'll need to open the file in a text editor and then create a new line after all the others. If you don't have a preferred editor, you can use the nano editor:

```
sudo nano /etc/fstab
```

For example, the `/etc/fstab` file after adding an entry for `/dev/sdb` would look similar to the following:

```

jay@ubuntu-server: ~
GNU nano 6.2 /etc/fstab *
# /etc/fstab: static file system information.
#
# Use 'blkid' to print the universally unique identifier for a
# device; this may be used with UUID= as a more robust way to name devices
# that works even if disks are added and removed. See fstab(5).
#
# <file system> <mount point> <type> <options> <dump> <pass>
# / was on /dev/ubuntu-vg/ubuntu-lv during curtin installation
/dev/disk/by-id/dm-uuid-LVM-dTs7ZjscXf0LLczXWK61mo7EuybQKBbKRmtCaIzd8gt56KLTNJKvOZ>
# /boot was on /dev/sda2 during curtin installation
/dev/disk/by-uuid/30912eb2-fac6-435b-b55b-0d94dafa36a1 /boot ext4 defaults 0 1
/swap.img none swap sw 0 0

# Extra storage
UUID=117910ed-0fb3-4ee9-9c40-db9473ee7e50 /mnt/extra_storage ext4 defaults 0 0

```

Figure 9.9: The `/etc/fstab` file after adding a new entry to it

In my example, I created a comment line with a little note about what the extra volume will be used for (Extra storage). It's always a good idea to leave comments, so other administrators will have a clue regarding the purpose of the extra storage. Then, I created a new line with the UUID of the volume, the mount point for the volume, the filesystem type, defaults option, and a dump/pass of 0 and 0.

The defaults option is one I've not mentioned before. By using defaults as your mount option in `fstab`, your mount will be given several useful options in one shot, without having to list them individually. Among the options included with defaults are the following, which are worth an explanation:

- `rw`: Device will be mounted read/write
- `exec`: Allow files within this volume to be executed as programs
- `auto`: Automatically mount the device at boot time
- `nouser`: Only root is able to mount the filesystem
- `async`: Output to the device should be asynchronous

Depending on your needs, the options included with defaults may or may not be ideal. Instead, you can call the options out individually, separated by commas, choosing only the ones you need. For example, with regard to `rw`, you may not want users to be allowed to change the content. In fact, I strongly recommend that you use `ro` (read-only) instead unless your users have a very strong use case for needing to make changes to files. I've actually learned this the hard way, where I've experienced an entire volume getting completely wiped out (and no one admitted to clearing the contents). This volume included some very important company data. From that point on, I mandated `ro` being used for everything, with a separate `rw` mount created, with only a select few (very responsible) people having access to it.

The `exec` option may also not be ideal. For example, if your volume is intended for storing files and backups, you may not want scripts to be run from that location. By using the inverse of `exec` (`noexec`), you can prevent scripts from running to create a situation where users are able to store files on the volume but not execute programs that are stored there.

Another option worth explaining is `auto`. The `auto` option basically tells your system to automatically mount that volume whenever the system boots or when you enter the following command:

```
sudo mount -a
```

When executed, `sudo mount -a` will mount any entry in your `/etc/fstab` file that has the `auto` option set. If you've used `defaults` as an option for the mount, those will be mounted as well since `defaults` implies `auto`. This way, you can mount all filesystems that are supposed to be mounted without rebooting your server (this command is safe to run whenever, as it will not disrupt anything that is already mounted).

The opposite of the `auto` option is `noauto`, which can be used instead. As you can probably guess by the name, an entry in `fstab` with the `noauto` option will not be automatically mounted and will not be mounted when you run `mount -a`. Instead, entries with this option will need to be mounted manually.

You may be wondering, then, what the point is of including an entry in `/etc/fstab` just to use `noauto`, which kind of seems to defeat the purpose. To explain this better, here is an example `fstab` entry with `noauto` being used:

```
UUID=e51bcc9e-45dd-45c7 /mnt/ext_disk ext4 rw,noauto 0 0
```

Here, let's say that I have an external disk that I only mount when I'm performing a backup. I wouldn't want this device mounted automatically at boot time (I may not always have it connected to the server), so I use the `noauto` option. But since I do have an entry for it in `/etc/fstab`, I can easily mount it any time after I connect it with the following command:

```
sudo mount /mnt/ext_disk
```

Notice that I didn't have to include the device name or options; only the destination path for the mount. The `mount` command knows what device I'm referring to since I have an entry in the `/etc/fstab` file for a device to be mounted at `/mnt/ext_disk`. This saves me from having to type the device name and options each time I want to mount the device. So, in addition to mounting devices at boot time, the `/etc/fstab` file also becomes a convenient place to declare devices that may be used on an on-demand basis but aren't always attached.

One final option I would like to cover before we move on is `users`. When used with a `mount` in `/etc/fstab`, this allows regular users (users other than `root`) to mount and unmount the filesystem. This way, `root` or `sudo` will not be necessary at all for a `mount` used with this option. Use this with care, but it can be useful if you have a device with non-critical data that you don't mind your users having full control over when mounting and unmounting.

While the concept of a text file controlling which devices are mounted on the system may seem odd at first, I think you'll appreciate being able to view a single file in order to find out everything that should be mounted and where it should be mounted. As long as administrators add all on-demand devices to this file, it can be a convenient place to get an overview of the filesystems that are in use on the server. As a bonus, you can also use the `mount` command (with no options) to have the system provide you with a list of everything that's mounted. Go ahead and try that, and I'll meet you in the next section.

Backing up and restoring volumes

Since we're dealing with servers, the data that's being stored on our storage devices is no doubt going to be extremely important. While it's normal to have a few test servers for use as test subjects in a typical environment, our servers usually exist to carry out a very important task. I can tell you from first-hand experience, never put too much trust in storage devices. In fact, I recommend not trusting them at all. I consider all storage to be temporary, as hard drives can and do break. If your important data is only stored on one device, it's not safe. In this section, I'm going to discuss some very important topics around backups.

First, consider RAID volumes. We haven't discussed them in this chapter because while the technology can still be beneficial, it's not as popular as it once was. Don't get me wrong, there's still a place for RAID, but it's just not as popular as it used to be. RAID allows you to join multiple disks in various configurations, which can result in a lower chance of losing data.

For example, RAID level 1 ensures that two hard disks always have the same data. If one of the disks physically fails, then you haven't actually lost anything. When you replace a failed disk in RAID, it will rebuild the array with the new disk and then you'll again benefit from having some expandability. RAID 5 allows you to have multiple disks to benefit from more space, and RAID 6 is the same but it allows you to have two disks fail before you lose data, rather than just one. Generally, that's the difference between one level of RAID and another; how many disks are allowed to fail before it becomes a problem.

However, RAID suffers from some serious problems. The worst is that it's *not* a backup solution. It doesn't advertise itself to be that, but many administrators mistakenly assume that their data is safe when utilizing RAID. The truth is, the level of protection RAID offers you is minimal. If there's a lightning storm and a power surge gets past your surge protector and fries a hard disk, chances are the other one will fry too. Generally, the environmental factors that cause one hard disk to fail will likely cause other disks to fail too. Worse yet, if a criminal breaks into your server room, grabs your server, and runs away with it, then the crook got away with your server *and* all the disks in your RAID anyway, so there are various scenarios where it won't save you. RAID can definitely be good to have, but it's more of a convenience than anything else.

Backups that are actually good exist off of the server somewhere else. The further away the backup is from the source server, the better. If you store your backups in a drawer outside of your server room, then that's certainly better than leaving an external backup disk connected all the time (which can also be susceptible to power surges just the same as an internal disk). But if a terrible storm takes out your entire building, then having the backup disk stored in the same physical location will work against you.

It may seem as though I'm being a bit overly dramatic here. But actually, I'm not. These situations can and do happen. Successful backups are resilient, and allow you to get your servers up and running quickly. Backups of your data are on a more important level than that, as some companies can go out of business if they lose their important files, which can include things like schematics that enable the company to be in business in the first place. As a system administrator, you'll need to develop a backup scheme that will account for as many scenarios as possible.

An effective backup routine will include several layers. Having an external disk is a useful backup, but why not have more than one, just in case one of them fails? Perhaps you can store one off-site, and swap the off-site and on-site backup disks weekly. In addition, perhaps you may use a command such as `rsync` to copy the files on your server to a remote server in another location periodically. You may even consider cloud backup solutions, which are another great addition.

In this section, I can't give you a backup scheme for your organization because the layout of your backup system will depend on the needs of the organization, which are different from one company to the next. But what I can leave you with are some tips:

- Make sure to test your backups regularly. Simply having a backup isn't enough—they have to actually work! Try to restore data from a backup periodically to test the effectiveness.
- Have at least three layers in your backup scheme, with at least one being off-site. This can be a combination of external hard disks, network-attached storage, cloud storage, mirroring data to another server in another location, or whatever makes the most sense for your organization.

- Consider encryption. Although it's beyond the scope of this chapter, if your backups fall into the wrong hands, then protected data may leak and be readable by people you don't want to have the information.
- Check the policies of your organization, and ensure that your backup scheme is compliant. Not all companies have such a scheme, but if yours does, this is critical. Consider retention (how long backups must be kept for) and how frequently backups must be updated. If you don't have organizational policies, consult a lawyer to determine whether there are legal retention requirements for the industry of your organization.

Above all else, the point is to keep your data safe. So far in this book, we've looked at creating additional volumes and mounting them, and we even took a quick look at `rsync` earlier on. You've already learned some of the tools that can be made a part of a backup scheme, and you'll learn about more methods before the book comes to a close. For now, keep these points in mind as you proceed through the book, and consider how each new skill you learn can be implemented as part of a backup scheme, if applicable.

LVM is one of my favorite technologies, giving us additional flexibility with our storage. In fact, let's take a look at that now.

Utilizing LVM

The needs of your organization will change with time. While we as server administrators always do our best to configure resources with long-term growth in mind, budgets and changes in policy always seem to get in our way. LVM is something that I'm sure you'll come to appreciate. In fact, technologies such as LVM are those things that make Linux the champion when it comes to scalability and cloud deployments. With LVM, you are able to resize your filesystems online, without needing to reboot your server.

Take the following scenario for example. Say you have an application running on a virtualized production server—a server that's so important that downtime would cost your organization serious money. When the server was first set up, perhaps you gave the application's storage directory a 100 GB partition, thinking it would never need more than that. Now, with your business growing, it's not only using a lot of space, but you're about to run out! What do you do? If the server was initially set up with LVM, you could add an additional storage volume, add it to your LVM pool, and grow your partition, all without rebooting your server! On the other hand, if you didn't use LVM, you're forced to find a maintenance window for your server and add more storage the old-fashioned way, which would include having it be inaccessible for a time.

With physical servers, you can install additional hard drives and keep them on standby without utilizing them to still gain the benefit of growing your filesystem online, even though your server isn't virtual. In addition, if your server supports hot-plugging, you can still add additional volumes without powering the server down.

It's for this reason that I must stress that you should always use LVM on storage volumes in virtual servers whenever possible. Let me repeat myself: you should *always* use LVM on storage volumes when you are setting up a virtual server! If you don't, this will eventually catch up with you when your available space starts to run out and you find yourself working over the weekend to add new disks.

This process might involve manually syncing data from one disk to another and then migrating your users to the new disk. This is not a fun experience, believe me. You might not think you'll be needing LVM right now, but you never know.

Getting started with LVM

When setting up a new server via Ubuntu's installer, you're given the option to use LVM during installation. But it's much more important for your storage volumes to use LVM, and by those, I mean the volumes where your users and applications will store their data. LVM is a good choice for your Ubuntu Server's root filesystem if you'd like the root filesystem to also benefit from the features of LVM. In order to get started with LVM, there are a few concepts that we'll need to understand, specifically **volume groups**, **physical volumes**, and **logical volumes**.

A volume group is a namespace given to all the physical and logical volumes that you wish to use with that implementation of LVM. Basically, a volume group is the highest name that encompasses your entire implementation of an LVM setup. Think of it as a kind of container that is able to contain disks. An example of this might be a volume group named `vg-accounting`. This volume group would be used as a location for the accounting department to store their files. It will encompass the physical volumes and logical volumes that will be in use by these users. It's important to note that you aren't limited to just a single volume group; you can have several, each with its own disks and volumes.

A physical volume is a physical or virtual hard disk that is a member of a volume group. For example, the hypothetical `vg-accounting` volume group may consist of three 100 GB hard disks, and each would be considered a physical volume. Keep in mind that these disks are still referred to as physical volumes in the context of LVM, even when the disks are virtual. Basically, any block device that is owned by a volume group is a physical volume.

Finally, logical volumes are similar in concept to partitions. Logical volumes can take up a portion, or the whole, of a disk, but unlike standard partitions, they may also span multiple disks. For example, a logical volume can include three 100 GB disks and be configured such that you would receive a cumulative total of 300 GB. When mounted, users will be able to store files there just as they would a single partition on a standard disk. When the volume gets full, you can add an additional disk and then grow the partition to increase its size. Your users would see it as a single storage area, even though it may consist of multiple disks.

The volume group can be named anything you'd like, but I always give mine names that begin with `vg-` and end with a name detailing its purpose. As I mentioned, you can have multiple volume groups. Therefore, you can have `vg-accounting`, `vg-sales`, and `vg-techsupport` (and so on) all on the same server. Then, you assign physical volumes to each. For example, you can add a 500 GB disk to your server and assign it to `vg-sales`. From that point on, the `vg-sales` volume group owns that disk. You're able to split up your physical volumes in any way that makes sense to you. Then, you can create logical volumes utilizing these physical volumes, which is what your users will use.

I think it's always best to work through an example when it comes to learning a new concept, so I'll walk you through such a scenario. In my case, I just created a local Ubuntu Server VM on my machine via VirtualBox, and then I added four additional 20 GB disks after I installed the distribution.

Virtualization is a good way to play around with learning LVM if you don't have a server available with multiple free physical disks.

To get started with LVM on a server that isn't already using it, you'll first need to have at least one additional (unused) volume, and install the required packages, which may or may not be present on your server. To find out if the required `lvm2` package is installed on your server, execute the following command:

```
apt search lvm2 |grep installed
```

If it's not present (the output of the previous command doesn't include `[installed,automatic]`), the following command will install the `lvm2` package and its dependencies:

```
sudo apt install lvm2
```

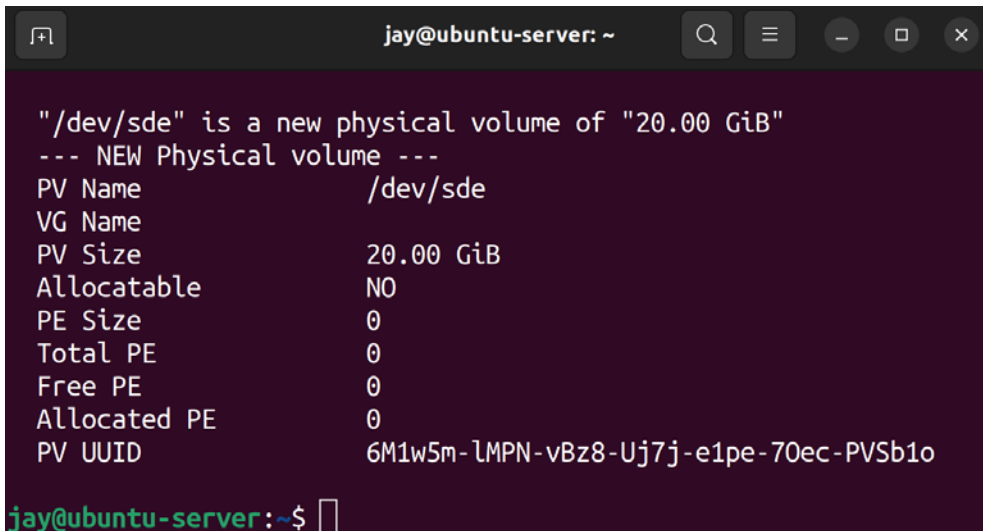
Next, we'll need to take an inventory of the disks we have available to work with. You can list them with the `fdisk -l` command as we've done several times now. In my case, I've added a few new volumes to my server, so now I have `/dev/sdb`, `/dev/sdc`, `/dev/sdd`, and `/dev/sde` to work with. The names of your disks will be different depending on your hardware or virtualization platform, so make sure to adjust all of the following commands accordingly.

To begin, we'll need to configure each disk to be used with LVM, by setting up each one as a physical volume. Note that we don't need to format a storage device, or even use `fdisk` to set it up before beginning the process of setting up LVM. Formatting actually comes later in this particular process. The `pvcreate` command is the first command we run to configure our disks for use with LVM. Therefore, we'll need to run the `pvcreate` command against all of the drives we wish to use for this purpose. For example, if I had four disks I wanted to use with LVM, I would run the following to set them up:

```
sudo pvcreate /dev/sdb
sudo pvcreate /dev/sdc
sudo pvcreate /dev/sdd
sudo pvcreate /dev/sde
```

And so on, for however many disks you plan on using.

To confirm that you have followed the steps correctly, you can use the `pvdisplay` command as root to display the physical volumes you have available on your server:



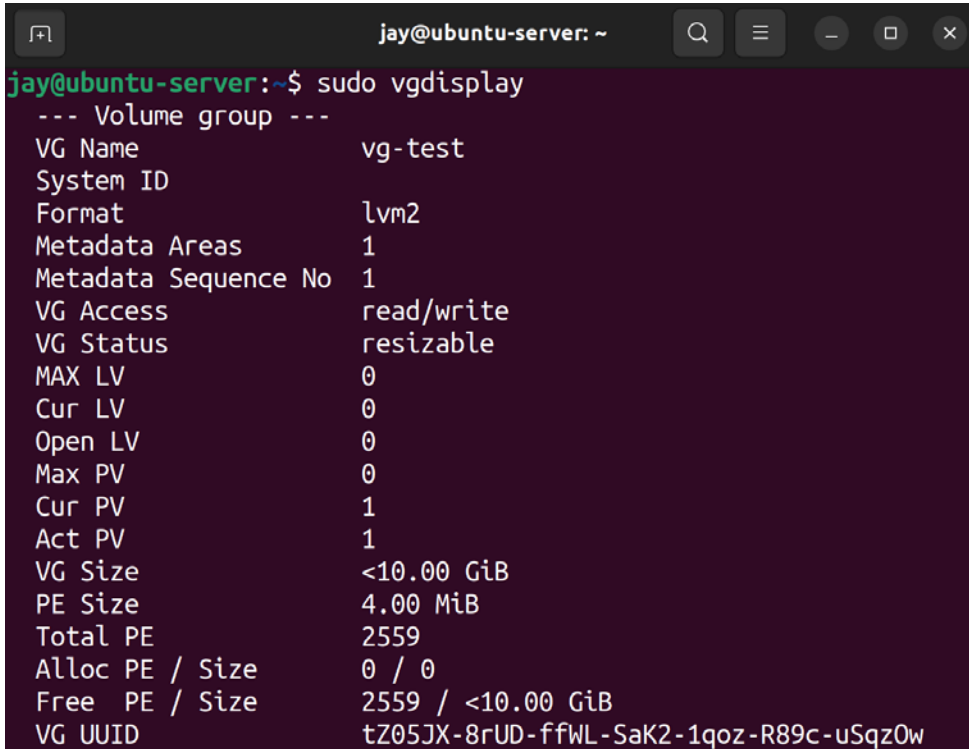
```
jay@ubuntu-server: ~  
"/dev/sde" is a new physical volume of "20.00 GiB"  
--- NEW Physical volume ---  
PV Name          /dev/sde  
VG Name  
PV Size          20.00 GiB  
Allocatable      NO  
PE Size          0  
Total PE         0  
Free PE          0  
Allocated PE     0  
PV UUID          6M1w5m-lMPN-vBz8-Uj7j-e1pe-70ec-PVSb1o  
jay@ubuntu-server:~$
```

Figure 9.10: Output of the `pvdisplay` command on a sample server

The screenshot shows only one volume, as it had to be formatted to fit this page. The `pvdisplay` command will show more output if you scroll up. Although we have some physical volumes to work with, none of them are assigned to a volume group. In fact, we haven't even created a volume group yet. We can now create our volume group with the `vgcreate` command, where we'll give our volume group a name and assign our first disk to it:

```
sudo vgcreate vg-test /dev/sdb
```

Here, I'm creating a volume group named `vg-test` and I'm assigning it one of the physical volumes I prepared earlier (`/dev/sdb`). Now that our volume group is created, we can use the `vgdisplay` command with `sudo` to view details about it, including the number of assigned disks (which should now be 1):



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ sudo vgdisplay
--- Volume group ---
VG Name                vg-test
System ID
Format                 lvm2
Metadata Areas         1
Metadata Sequence No   1
VG Access               read/write
VG Status               resizable
MAX LV                 0
Cur LV                 0
Open LV                 0
Max PV                  0
Cur PV                 1
Act PV                  1
VG Size                 <10.00 GiB
PE Size                 4.00 MiB
Total PE                2559
Alloc PE / Size         0 / 0
Free PE / Size          2559 / <10.00 GiB
VG UUID                 tZ05JX-8rUD-ffWL-SaK2-1qoz-R89c-uSqz0w

```

Figure 9.11: Output of the `vgdisplay` command on a sample server

At this point, if you created four virtual disks as I have, you have three more disks left that are not part of the volume group. Don't worry, we'll come back to them later. Let's forget about them for now as there are other concepts to work on at the moment.

All we need to do at this point is to create a logical volume and format it. Our volume group can contain all of, or a portion of, the disk we've assigned to it. With the following command, I'll create a logical volume of 5 GB from the virtual disk I added to the volume group:

```
sudo lvcreate -n myvol1 -L 5g vg-test
```

The command may look complicated, but it's not. In this example, I'm giving my logical volume the name `myvol1` with the `-n` option. Since I only want to give it 5 GB of space, I use the `-L` option and then `5g` to represent 5 GB. Finally, I give the name of the volume group that this logical volume will be assigned to. You can run `lvdisplay` with `sudo` to see information regarding this volume:

```
jay@ubuntu-server: ~  
jay@ubuntu-server:~$ sudo lvsdisplay  
--- Logical volume ---  
LV Path                /dev/vg-test/myvol1  
LV Name                 myvol1  
VG Name                 vg-test  
LV UUID                 y0R0c1-CX6z-a55I-Wktp-1Jpi-Pvea-zczNPv  
LV Write Access         read/write  
LV Creation host, time ubuntu-server, 2022-05-09 01:20:58 +0000  
LV Status                available  
# open                  0  
LV Size                 5.00 GiB  
Current LE              1280  
Segments                1  
Allocation              inherit  
Read ahead sectors     auto  
- currently set to     256  
Block device            253:1
```

Figure 9.12: Output of the `lvsdisplay` command on a sample server

At this point, we should have everything we need as far as setting up LVM is concerned. But we still need to format a volume before we can use it, similar to a non-LVM disk.

Formatting logical volumes

Next, we need to format our logical volume so that it can be used. However, as always, we need to know the name of the device so that we know what it is we're formatting. With LVM this is easy. The `lvsdisplay` command gave us this already; you can see it in the output (it's the third line down in *Figure 9.12*, under `LV Path`). Let's format it with the `ext4` filesystem:

```
sudo mkfs.ext4 /dev/vg-test/myvol1
```

And now this device can be mounted as any other hard disk. I'll mount mine at `/mnt/lvm/myvol1`, but you can use any directory name you wish:

```
sudo mount /dev/vg-test/myvol1 /mnt/lvm/myvol1
```

To check our work, execute `df -h` to ensure that our volume is mounted and shows the correct size. We now have an LVM configuration containing just a single disk, so this isn't very useful. The 5 GB I've given it will not likely last very long, but there is some remaining space we can use that we haven't utilized yet. With the following `lvextend` command, I can resize my logical volume to take up the remainder of the physical volume:

```
sudo lvextend -n /dev/vg-test/myvol1 -l +100%FREE
```

In this case, `+100%FREE` is the argument that clarified that we are wanting to use the entirety of the remaining space for the logical volume. If done correctly, you should see output similar to the following:

```
Logical volume vg-test/myvol1 successfully resized.
```

Now my logical volume is using the entire physical volume I assigned to it. Be careful, though, because if I had multiple physical volumes assigned, that command would've claimed all the space on those as well, giving the logical volume a size that is the total of all the space it has available, across all its disks. You may not always want to do this, but since I only had one physical volume anyway, I don't mind. Go ahead and check your free space again with the `df -h` command:

```
df -h
```

Unfortunately, it's not showing the extra space we've given the volume. The output of `df` is still showing the size the volume was before. That's because although we have a larger logical volume, and it has all the space assigned to it, we didn't actually resize the ext4 filesystem that resides on this logical volume. To do that, we will use the `resize2fs` command:

```
sudo resize2fs /dev/mapper/vg--test-myvol1
```

The double-hyphen in the previous command is intentional, so make sure you're typing the command correctly.

If run correctly, you should see output similar to the following:

```
The filesystem on /dev/mapper/vg--test-myvol1 is now 5241856 (4k) blocks long.
```

Now you should see the added space as usable when you execute `df -h`. The coolest part is that we resized an entire filesystem without having to restart the server. In this scenario, if our users have got to the point where they have utilized the majority of their free space, we will be able to give them more space without disrupting their work.

However, you may have additional physical volumes that have yet to be assigned to a volume group. In my example, I created four and have only used one in the LVM configuration so far. We can add additional physical volumes to our volume group with the `vgextend` command. In my case, I'll run this against the three remaining drives. If you have additional physical volumes, feel free to add yours with the same commands I use, but substitute my device names with yours:

```
sudo vgextend vg-test /dev/sdc
sudo vgextend vg-test /dev/sdd
sudo vgextend vg-test /dev/sde
```

You should see a confirmation similar to the following:

```
Volume group "vg-test" successfully extended
```

When you run `pvdiskdisplay` now, you should see the additional physical volumes attached that weren't showing there before. Now that we have extra disks in our LVM configuration, we have some additional options.

We could give all the extra space to our logical volume right away and extend it as we did before. However, I think it's better to withhold some of the space from our users. That way, if our users do use up all our available space again, we have an emergency reserve of space we could use at a pinch if we needed to while we figure out the long-term solution. In addition, LVM snapshots (which we will discuss soon) require you to have unallocated space in your LVM setup.

The following example command will add an additional 10 GB to the logical volume:

```
sudo lvextend -L+10g /dev/vg-test/myvol1
```

And finally, make the free space available to the filesystem:

```
sudo resize2fs /dev/vg-test/myvol1
```

With very large volumes, the resizing may take some time to complete. If you don't see the additional space right away, you may see it gradually increase every few seconds until all the new space is completely allocated.

Removing volumes with LVM

Finally, you may be curious about how to remove a logical volume or volume group. For these purposes, you would use the `lvremove` or `vgremove` commands. It goes without saying that these commands are destructive, but they are useful in situations where you want to delete a logical volume or volume group. To remove a logical volume, the following syntax will do the trick:

```
sudo lvremove vg-test/myvol1
```

Basically, all you're doing is giving the `lvremove` command the name of your volume group, a forward slash, and then the name of the logical volume within that group that you would like to remove. To remove the entire volume group, the following command and syntax should be fairly self-explanatory:

```
sudo vgremove vg-test
```

You can only remove a logical volume if it's not in use, and this may not be something you'll do very often, but if you ever do need to decommission an LVM component, then there are commands that will enable you to do so.

Hopefully, you're convinced by now how awesome LVM is. It allows you flexibility over your server's storage that other platforms can only dream of. The flexibility of LVM is one of the many reasons why Linux excels in the cloud market. These concepts can be difficult to grasp at first if you haven't worked with LVM before. But thanks to virtualization, playing around with LVM is easy. I recommend you practice creating, modifying, and destroying volume groups and logical volumes until you get the hang of it. If the concepts aren't clear now, they will be with practice.

In this section, you saw some ways in which LVM can benefit you; it allows you to take the storage of your server to the next level, even expanding it and growing it on demand. However, LVM also has additional tricks up its sleeve. It even allows you to create snapshots as well. We'll cover this useful ability next.

Understanding LVM snapshots

LVM snapshots allow you to capture a logical volume at a certain point in time and preserve it. After you create a snapshot, you can mount it as you would any other logical volume and even revert your volume group to the snapshot if something fails. In practice, this is useful if you want to test some potentially risky changes to files stored within a volume, but want the insurance that if something goes wrong, you can always undo your changes and go back to how things were. LVM snapshots allow you to do just that. LVM snapshots require you to have some unallocated space in your volume group.

However, LVM snapshots are definitely *not* a viable form of backup. For the most part, these snapshots are best when used as a temporary holding area when running tests or testing out experimental software before rolling out changes to production systems. During Ubuntu's installation process, you were offered the option to create an LVM configuration. Therefore, you can use snapshots to test how security updates will affect your server if you used LVM for your root filesystem. If the new updates start to cause problems, you can always revert back. When you're done testing, you should merge or remove your snapshot.

So, why did I refer to LVM snapshots as a temporary solution and not a backup? First, similar to our discussion earlier, backups aren't secure if they are stored on the same server that's being backed up. It's always important to save backups of the server at least, preferably off-site. But what's worse is that if your snapshot starts to use up all available space in your volume group, it can get corrupted and stop working. Therefore, this is a feature you would use with caution, just as a means of testing something, and then revert back or delete the snapshot when you're done experimenting. Don't leave an LVM snapshot hanging around for too long.

When you create a snapshot with LVM, what happens is a new logical volume is created that is a clone of the original. Initially, no space is consumed by this snapshot. But as you run your server and manipulate files in your volume group, the original blocks are copied to the snapshot as you change them, to preserve the original logical volume. If you don't keep an eye on usage, you may lose data if you aren't careful and the logical volume will fill up.

To show this in an example, the following command will create a snapshot (called `mynsnapshot`) of the `myvol1` logical volume:

```
sudo lvcreate -s -n mynsnapshot -L 4g vg-test/myvol1
```

You should see the following output:

```
Logical volume "mynsnapshot" created.
```

With that example, we're using the `lvcreate` command, with the `-s` option (snapshot) and the `-n` option (which allows us to name the snapshot), where we declare a name of `mynsnapshot`. We're also using the `-L` option to designate a maximum size for the snapshot, which I set to 4 GB in this case. Finally, I give it the volume group and logical volume name, separated by a forward slash (/). From here, we can use the `lvs` command to monitor its size.

Since we're creating a new logical volume when we create a snapshot, we can mount it as we would a normal logical volume. This is extremely useful if we want to pull a single file without having to restore the entire thing.

But what about restoring the snapshot? One of the major benefits of snapshots is the ability to “roll back” to when the snapshot was taken. Essentially, this allows you to test changes to the server and then undo those changes. To roll back to a snapshot, we can do so with the `lvconvert` command:

```
sudo lvconvert --merge vg-test/mysnapshot
```

The output will look similar to the following:

```
Merging of volume mysnapshot started.  
myvol1: Merged: 100.0%
```

However, it's important to note, that, unlike being able to resize a logical volume online, we cannot merge a snapshot while it is in use. If you do, the changes will take effect the next time it is mounted. Therefore, you can either unmount the logical volume before merging or unmount and remount after merging. Afterward, you'll see that the snapshot is removed the next time you run the `lvs` command.

Since you cannot merge (roll back) a snapshot that is in use, if the snapshot is of the root filesystem, you'll have to reboot the server for the rollback to finalize.

If you'd like to make the snapshot permanent, which finalizes all of the changes you've made since the snapshot was first taken, we can use the `lvremove` command. For our example snapshot in this section, we can use the following command to make the snapshot permanent:

```
sudo lvremove vg-test/mysnapshot
```

As you can probably conclude based on the name of the command, `lvremove` deletes the snapshot. The act of deleting the snapshot is actually what makes its changes final, while the `lvconvert` command mentioned earlier rolls back to when the snapshot was taken.

LVM snapshots are definitely a useful feature, even if it's not supposed to be considered a backup solution. My favorite use case for these snapshots is to take a snapshot of the root filesystem before installing all available updates. After I reboot, and the updates take effect, I can either delete the snapshot (if everything seems to be fine) or revert back to the snapshot if the updates seem to be causing a problem. If nothing else, LVM snapshots are yet another trick you can use when and if the need for it comes up.

Summary

Efficiently managing the storage of your servers will ensure that things continue to run smoothly, as a full filesystem can definitely cause your server to grind to a halt. Thankfully, Linux servers feature a very expansive toolset for managing your storage, some of which are a source of envy for other platforms. As Linux server administrators, we benefit from technologies such as LVM and utilities such as `ncdu`, as well as many others. In this chapter, we explored these tools and how to manage our storage. We covered how to format, partition, mount, and unmount volumes, as well as manage the `fstab` file, LVM, monitor disk usage, and more.

In the next episode of our Ubuntu Server saga, we'll work through connecting to networks. We'll configure our server's hostname, work through examples of connecting to other servers via OpenSSH, and take a look at IP addressing.

Relevant videos

- Linux Crash Course – fstab: <https://linux.video/fstab>
- Linux EssentialsCrash Course – Formatting and mounting storage volumes: <https://linux.video/format-mount>
- LVM deep dive: <https://linux.video/lvm>

Further reading

- Ubuntu LVM documentation: <https://learnlinux.link/u-lvm>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaz0>



10

Connecting to Networks

Linux networks took the IT industry by storm. Many organizations use Linux in their data centers, on both physical servers and in the cloud. Ubuntu Server is among the most popular choices for running mission-critical applications, but without a stable network to connect the individual components of your infrastructure together, even the most powerful server hardware will be ineffective.

So far in this book, we've worked with a single Ubuntu Server instance. Here, we begin a two-part look at networking in Linux. In this chapter, we'll discuss topics related to initial network connectivity and remote management. We'll continue learning additional networking topics in *Chapter 11, Setting Up Network Services*, where we'll work on building and configuring additional components that will enable your servers to communicate more effectively, which will result in a strong foundational network that will serve your needs for years to come.

In this episode of our Ubuntu adventure, we will cover:

- Setting the hostname
- Managing network interfaces
- Assigning static IP addresses
- Understanding Linux name resolution
- Getting started with OpenSSH
- Getting started with SSH key management
- Simplifying SSH connections with a config file

To get started in our exploration of networking, we should first give each of our Ubuntu servers its own identity; basically, we should give them a name to help distinguish each from the others.

Setting the hostname

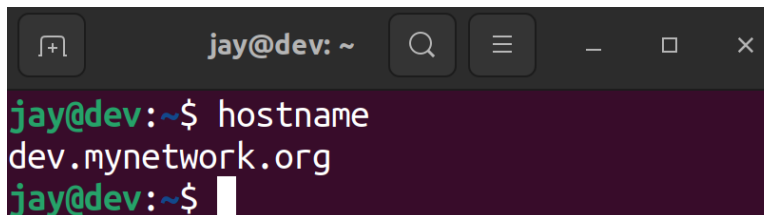
During installation, you were asked to create a hostname for your server. Specifically, the field was labeled `Your server's name` during the initial setup process. At that time, our goal was to simply get an Ubuntu Server installation set up for working through the examples in this book. At this point, you may consider changing the hostname of your server.

When we utilize OpenSSH to remotely manage our servers (as we'll do later on in this chapter), the hostname is shown on the command line. That can be very confusing if all servers have the same name. More importantly, the hostname of a server gives it an identity. When it comes to real production deployments of Ubuntu Server, each individual server should have its own designated purpose, and be named accordingly. Often, organizations will have their own naming scheme. Perhaps web servers in a company are named similar to `webserver-01`, or with a fully qualified domain name, such as `webserver-01.example.com`.

In this book, I won't assume any particular naming scheme, so when we do work through changing the hostname, feel free to adjust the name as you see fit. If you don't have a naming scheme (but would like to create one) feel free to get creative. I've seen quite a few variations, from naming servers after cartoon characters (who wouldn't want a server named `daffy-duck`?), to Greek gods or goddesses. Some companies choose to be a bit boring and come up with naming schemes consisting of a series of characters separated by hyphens, with codes representing which rack the server is in, as well as its purpose. You can create your own naming convention if you haven't already, and no matter what you come up with, I won't judge you.

As I mentioned, the hostname of your server is its identity. It identifies your server to the rest of the network. While a simple hostname, such as `ubuntu`, is fine if you have just one host, it would get confusing really quickly if you kept the default on every Ubuntu Server within your network. Giving each server a descriptive name helps you tell them apart from each other. But there's more to a server's name than its hostname, which we'll get into in *Chapter 11, Setting Up Network Services*, when we discuss DNS. But for now, we'll work through viewing and configuring the hostname, so you'll be ready to make your hostname official with a DNS assignment, when we come to it.

So, how do you view your hostname? One way is to simply look at your shell prompt; you've probably already noticed that your hostname is included there. While you can customize your shell prompt in many different ways, the default shows your current hostname. However, depending on what you've named your server, it may or may not show the entire name. Basically, the default prompt (known as a **PS1 prompt**, in case you were wondering) shows the hostname only until it reaches the first period. For example, if your hostname is `dev.mynetwork.org`, your prompt will only show `dev`. To view the entire hostname, simply enter the `hostname` command:

A terminal window with a dark background. The prompt is `jay@dev: ~`. The user has entered `hostname` and the output is `dev.mynetwork.org`. The prompt is now `jay@dev: ~$` with a cursor.

```
jay@dev: ~$ hostname
dev.mynetwork.org
jay@dev: ~$
```

Figure 10.1: Output from the `hostname` command

Changing the hostname is fairly simple. To do this, we can use the `hostnamectl` command as root or with `sudo`. If, for example, I'd like to change my hostname from `dev.mynetwork.org` to `dev2.mynetwork.org`, I would execute the following command:

```
sudo hostnamectl set-hostname dev2.mynetwork.org
```

Simple enough, but what does that command actually do? Well, I'd love to give you a fancy outline, but all it really does is change the contents of a text file (specifically, `/etc/hostname`). To see this for yourself, feel free to use the `cat` command to view the contents of this file before and after making the change with `hostnamectl`:

```
cat /etc/hostname
```

You'll see that this file contains only your hostname.

Once you change your hostname, you may start seeing an error message similar to the following after executing some commands:

```
unable to resolve host dev.mynetwork.org
```

This error means that the computer is no longer able to resolve your local hostname. This is due to the fact that the `/etc/hostname` file is not the only file where your hostname is located; it's also referenced in `/etc/hosts`. Unfortunately, the `hostnamectl` command doesn't update `/etc/hosts` for you, so you'll need to edit that file yourself to make the error go away. Here's what an `/etc/hosts` file looks like on an example server:

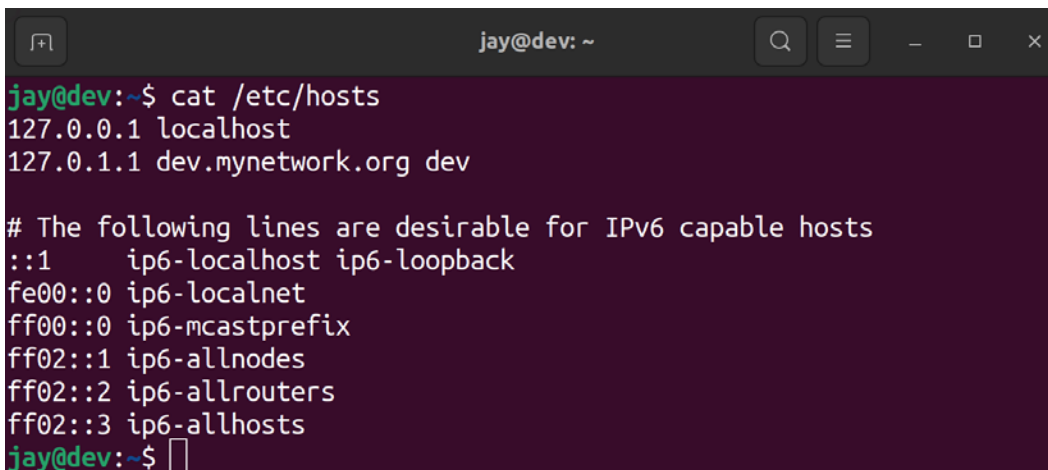
A terminal window with a dark purple background. The prompt is 'jay@dev: ~'. The user has entered 'cat /etc/hosts'. The output shows: '127.0.0.1 localhost', '127.0.1.1 dev.mynetwork.org dev', a comment line '# The following lines are desirable for IPv6 capable hosts', and several IPv6 addresses: '::1 ip6-localhost ip6-loopback', 'fe00::0 ip6-localnet', 'ff00::0 ip6-mcastprefix', 'ff02::1 ip6-allnodes', 'ff02::2 ip6-allrouters', and 'ff02::3 ip6-allhosts'. The prompt returns to 'jay@dev:~\$' with a cursor.

Figure 10.2: Sample contents from an `/etc/hosts` file

The first two entries, in this example, refer to the local machine itself. Localhost addresses, also known as **loopback addresses**, allow the machine to essentially reach itself. If you were to use the `ping` command against the `127.0.0.1` address, the reply would come from the machine that executed the command, not from another host on the network. On the first line, we have the following:

```
127.0.0.1 localhost
```

If you were to use any networking command to attempt to communicate with the local server, such as pinging `127.0.0.1` or `localhost`, the `/etc/hosts` file on this line declares that this communication is directed toward the underlying server itself.

With the second line in the example screenshot, we have the following:

```
127.0.1.1 dev.mynetwork.org dev
```

Depending on your configuration, such as whether you are using a physical server, a virtualization platform, or a cloud server provider, that line may or may not be present. You can add that line if it's missing, but we'll talk about that more in a moment.

Essentially, that particular line identifies that the local server can also be reached at the IP address `127.0.1.1`, the fully qualified domain name of `dev.mynetwork.org`, as well as the simplified form of `dev`. A fully qualified domain name consists of the name of the server (`dev` in this case) as well as the domain name for the organization (`mynetwork.org` in this example). This enables you to ping your local server directly from that server by using the name `dev.mynetwork.org` or the simplified form of `dev`.

If you don't have a domain name to use with your servers, you can leave the fully qualified domain name out of the `/etc/hosts` file. So in our example, that line would look like the following if there was no domain:

```
127.0.1.1 dev
```

Going back to our example of changing a hostname on a server, I mentioned that you can use the `hostnamectl` command to do that, but that command doesn't update the `/etc/hosts` file for you, it only updates the `/etc/hostname` file. It's a best practice to also update the `/etc/hosts` file to match. You can avoid using the `hostnamectl` command altogether and manually edit the `/etc/hosts` and `/etc/hostname` files, which is actually my preferred method. If I have to manually edit a text file, regardless of whether or not I use the `hostnamectl` command, I figure that I may as well use a text editor for both.

The main takeaway, though, is to give your servers an identity that makes sense and matches the role that the server will fill within your network. At a typical organization, you'll have web servers, file servers, database servers, and more. A consistent and logical naming scheme will just make everything that much easier.

Now that we have learned how to give our servers an identity, we can learn how to manage network interfaces.

Managing network interfaces

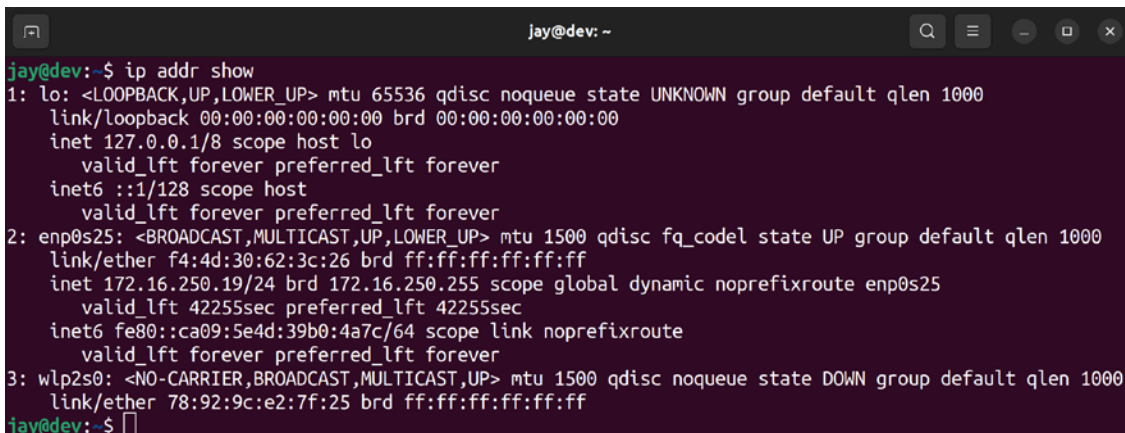
Networking is critical for server infrastructure. Without a network, servers cannot communicate with one another, and users will be unable to access them. In order for a server to connect to a network, it needs to have a network interface installed. Most servers will have a standard wired Ethernet adapter installed, allowing you to plug in a network cable to connect it to a switch. Assuming our server's hardware has been properly detected by Ubuntu, this is handled pretty much automatically. However, the automatic configuration is not always ideal. Perhaps we want to customize the IP address or settings related to the connection.

First, we need to understand how to view the current connection parameters that the network card of our server currently has in effect. That's the main goal of this section. We can do so using two basic commands: `ip` (which is recommended) and `ifconfig` (which was the previous method in older versions of Ubuntu, and its usage is no longer recommended).

We can review information regarding our network interfaces and manage them with the `ip` command. For example, we can use `ip addr show` to view our currently assigned IP address:

```
ip addr show
```

This will produce an output similar to that shown in the following screenshot:



```
jay@dev: ~
jay@dev:~$ ip addr 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
2: enp0s25: <BROADCAST,MULTICAST,UP,LOWER_UP> mtu 1500 qdisc fq_codel state UP group default qlen 1000
    link/ether f4:4d:30:62:3c:26 brd ff:ff:ff:ff:ff:ff
    inet 172.16.250.19/24 brd 172.16.250.255 scope global dynamic noprefixroute enp0s25
        valid_lft 42255sec preferred_lft 42255sec
    inet6 fe80::ca09:5e4d:39b0:4a7c/64 scope link noprefixroute
        valid_lft forever preferred_lft forever
3: wlp2s0: <NO-CARRIER,BROADCAST,MULTICAST,UP> mtu 1500 qdisc noqueue state DOWN group default qlen 1000
    link/ether 78:92:9c:e2:7f:25 brd ff:ff:ff:ff:ff:ff
jay@dev:~$
```

Figure 10.3: Viewing IP information with the `ip addr show` command

Once you enter that command, you should see output that pertains to the network interfaces that you have available and their current status. Also, you can abbreviate the command all the way down to simply `ip a` (the output will be the same in either case). From the output, we can see several useful tidbits, such as the IP address for each device (if it has one), as well as its MAC address.

Using the `ip` command, we can also manage the state of an interface. We can bring a device down (prevent it from connecting to networks), and then back up again:

```
sudo ip link set enp0s3 down
sudo ip link set enp0s3 up
```

In that example, I'm simply toggling the state for interface `enp0s3`. First, I'm bringing it down, and then I'm bringing it back up again.

Bringing interfaces up and down is all well and good, but what's up with that naming convention? The convention used in Ubuntu 22.04 may seem a bit strange for those of you that have grown accustomed to the scheme used in earlier versions, which utilized network interface names such as `eth0`, `wlan0`, and so on. Since Ubuntu is based on Debian, it has adopted the new naming convention that was introduced starting with Debian 9.0.

The new naming convention has been put in place in order to make interface naming more predictable. While you may argue that names such as `eth0` may be easier to memorize than something like `enp0s3`, the change helps the name stay persistent between boots. When you add new network interfaces to a Linux system, there's always the possibility that other interface names may change as well.

For example, if you have an older Linux installation on a server with a single network card (`eth0`) and you add a second (which is given the name `eth1`), your configuration may break if the names were to get switched during the next boot. Imagine for a moment that one interface is connected to the internet and another connected to a switch (basically, you have an internet gateway). If the interfaces came up in the wrong order, internet access would be disrupted for your entire office, due to the fact that the firewall rules you've written are being applied to the wrong interfaces. Definitely not a pleasant experience!

In the past, previous versions of Ubuntu (as well as Debian, and even CentOS) have opted to use `udev` to make the names stick in order to work around this issue. This is no longer necessary nowadays, but I figured I'd mention it here just in case you end up working on a server with an older installation. These older servers would achieve stickiness with interface names from configuration stored in the following file:

```
/etc/udev/rules.d/70-persistent-net.rules
```

This file existed on older versions of some popular Linux distributions (including Ubuntu), as a workaround to this problem. This file contains some information that identifies specific qualities of the network interface, so that with each boot, it will always come up with the same name. Therefore, the card you recognize as `eth0` will always be `eth0`. If you have an older version of Ubuntu Server in use, you should be able to see this file for yourself. Here's some sample output of this file on an older installation:

```
SUBSYSTEM=="net", ACTION=="add", DRIVERS=="?*",  
TTR{address}=="01:22:4e:a5:f2:ec", ATTR{dev_id}=="0x0", ATTR{type}=="1",  
KERNEL=="eth*", NAME="eth0"
```

As you can see, it's using the MAC address of the card to identify it with `eth0`. But this becomes a small problem if I want to take an image of this machine and re-deploy it onto another server. This is a common practice—we administrators rarely start over from scratch if we don't have to, and if another server is similar enough to a new server's desired purpose, cloning it will be an option. However, when we restore the image onto another server, the `/etc/udev/rules.d/70-persistent-net.rules` file will come along for the ride. We'll more than likely find that the new server's first network interface will have a designation of `eth1`, even if we only have one interface.

This is because the file already designated a device as `eth0` (it's referencing a device that's not present in the system), so we would need to correct this file ourselves in order to reclaim `eth0`. We would do that by editing the `rules` file, deleting the line that contains the card that's not on the system, and then changing the device designation on the remaining line back to `eth0`.

The new naming scheme is effective as of `systemd v197` and later (in case you didn't already know from earlier topics in this book, `systemd` is the underlying framework utilized in Ubuntu for managing processes and various resources). For the most part, the new naming convention references the physical location of the network card on your system's bus.

Therefore, the name it receives cannot change unless you were to actually remove the network card and place it in a different slot on the system's board, or change the position of the virtual network device in your hypervisor.

As a quick overview of how the network names break down, `en` is for Ethernet, and `wl` is for wireless. Therefore, we know that the example interface I mentioned earlier (`enp0s3`) references a wired card. The `p` references which bus is being used, so `p0` refers to the system's first PCI bus (the numbering starts at 0). Next, we have `s3`, which references PCI slot 3. Putting it together, `enp0s3` references a wired network interface card on the system's first bus, placed in PCI slot 3. The exact details of the new naming specification could even be a chapter of its own, but hopefully this gives you a general idea of how the new naming convention breaks down. There's much more documentation online if you're interested in the nitty-gritty details (see the *Further reading* section). The important point here is that since the new naming scheme is based on where the card is physically located, it's much less likely to change abruptly. In fact, it can't change, as long as you don't physically switch the positions of your network cards inside the case.

Getting back to managing our interfaces, another command worth discussion is `ifconfig`.

The `ifconfig` command is part of the `net-tools` suite of utilities, which has been deprecated (for the most part). Its replacement is the `iproute2` suite of utilities, which includes the `ip` command we've already discussed. In summary, this basically means you should be using commands from the `iproute2` suite, instead of commands such as `ifconfig`. The problem, though, is that most administrators nowadays still use `ifconfig`, with no sign of it slowing down. In fact, the `net-tools` suite has been recommended for deprecation for years now, and many Linux distributions shipping today still have this suite installed by default. Those that don't have it installed offer it as an additional package that you can install. In the case of Ubuntu Server 22.04, the `net-tools` package is no longer installed by default, but it's still available if you want to manually install it. I don't recommend installing it though, since it's deprecated and shouldn't be used anymore.

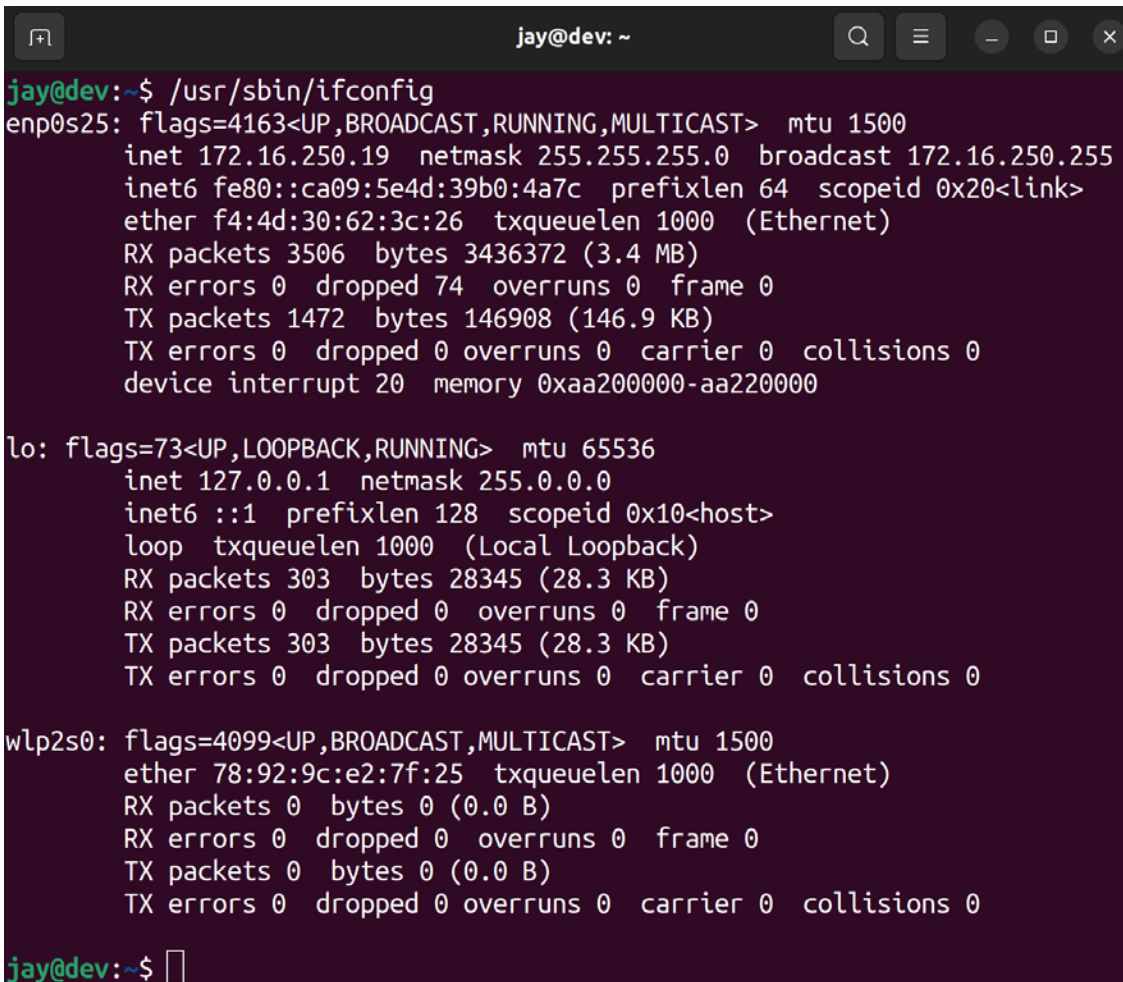
The reason commands such as `ifconfig` have a tendency to stick around so long after they've been deprecated usually comes down to the *change is hard* mentality, but quite a few scripts and programs out there are still using `ifconfig`, and therefore it's worth discussing here. Even if you immediately stop using `ifconfig`, and move to `ip` from now on, you'll still encounter this command on your travels, so you may as well know a few examples. Knowing the older commands will also help you if you find yourself on an older server.

First, when executed by itself with no options, `ifconfig` will print information regarding your interfaces like we did with `ip addr show` earlier. That seems pretty simple.

If you are unable to use `ifconfig` to view interface information using a normal user, try using the fully qualified command (include the full path):

```
/usr/sbin/ifconfig
```

The `/usr/sbin` directory may or may not be in your `$PATH` (a set of directories your shell looks within for commands), so if your system doesn't recognize `ifconfig`, using the fully qualified command should produce the desired output, as follows:



```
jay@dev:~$ /usr/sbin/ifconfig
enp0s25: flags=4163<UP,BROADCAST,RUNNING,MULTICAST> mtu 1500
    inet 172.16.250.19 netmask 255.255.255.0 broadcast 172.16.250.255
    inet6 fe80::ca09:5e4d:39b0:4a7c prefixlen 64 scopeid 0x20<link>
    ether f4:4d:30:62:3c:26 txqueuelen 1000 (Ethernet)
    RX packets 3506 bytes 3436372 (3.4 MB)
    RX errors 0 dropped 74 overruns 0 frame 0
    TX packets 1472 bytes 146908 (146.9 KB)
    TX errors 0 dropped 0 overruns 0 carrier 0 collisions 0
    device interrupt 20 memory 0xaa200000-aa220000

lo: flags=73<UP,LOOPBACK,RUNNING> mtu 65536
    inet 127.0.0.1 netmask 255.0.0.0
    inet6 ::1 prefixlen 128 scopeid 0x10<host>
    loop txqueuelen 1000 (Local Loopback)
    RX packets 303 bytes 28345 (28.3 KB)
    RX errors 0 dropped 0 overruns 0 frame 0
    TX packets 303 bytes 28345 (28.3 KB)
    TX errors 0 dropped 0 overruns 0 carrier 0 collisions 0

wlp2s0: flags=4099<UP,BROADCAST,MULTICAST> mtu 1500
    ether 78:92:9c:e2:7f:25 txqueuelen 1000 (Ethernet)
    RX packets 0 bytes 0 (0.0 B)
    RX errors 0 dropped 0 overruns 0 frame 0
    TX packets 0 bytes 0 (0.0 B)
    TX errors 0 dropped 0 overruns 0 carrier 0 collisions 0

jay@dev:~$
```

Figure 10.4: Viewing interface information with the `ifconfig` command

Secondly, just like with the `ip` commands we practiced earlier, we can also bring an interface down or up with `ifconfig` as well:

```
sudo ifconfig enp0s3 down
sudo ifconfig enp0s3 up
```

There are, of course, other options and variations of `ip` and `ifconfig`, so feel free to look up the man pages for either if you want more information. For the purposes of this section, the main thing is to remember how to view your current IP assignments, as well as how to bring an interface up or down.

As useful as our network interfaces can be, they're useless without an IP address assigned to them. While a network will often use DHCP to take care of this, in the next section we'll take a look at how to assign a static IP address.

Assigning static IP addresses

With servers, it's very important that your IP addresses remain fixed and do not change for any reason. If an IP address does change (such as a dynamic lease with no reservation), your users will experience an outage, services will fail, or entire sites may become unavailable. When you install Ubuntu Server, it will grab a dynamically assigned lease from your DHCP server, but after you configure the server the way you want it, it's important to set a permanent IP address right away before it's considered production-ready. One exception to this rule is an Ubuntu-based VPS. Cloud providers that bill you for these servers will have an automatic system in place to declare an IP address for your new VPS, and will already have it configured to remain in place. But in the case of virtual or physical servers you manage yourself, you'll start off with a dynamic address, unless you've already configured a static IP address during installation.

In most cases, you'll have an IP address scheme in place at your office or organization, which will outline a range of IP addresses that are available for use with static assignments. If you don't have such a scheme, it's important to create one, so you will have less work to do later when you bring more servers online. We'll talk about setting up a DHCP server and IP address scheme in *Chapter 11, Setting Up Network Services*, but for now, I'll give you a few quick tips. Your DHCP server will typically have a range of IP addresses that will be automatically assigned to any host that requests an assignment. When setting up a static IP on a server, you'll want to make sure that the IP address that you choose is outside of the range that your DHCP server assigns so you don't end up with a duplicate IP on your network. For example, if your DHCP server assigns IPs ranging from `10.10.10.100` through `10.10.10.150`, you'll want to use an IP address *not* included within that range for your servers.

There are two ways of assigning a fixed address to a network host, including your servers. The first is by using a static IP assignment, as I've already mentioned. With that method, you'll arbitrarily grab an IP address that's not being used by anything, and then configure your Ubuntu Server to use that address. In that case, your server is never requesting an IP address from your network's DHCP server. It simply uses whatever you assign it. This is the method I'll be going over in this section.

The other way of assigning a fixed address to a server is by using a static lease. This is also known as a **DHCP reservation**, but I prefer to use the former term. With this method, you configure your DHCP server to assign a specific IP address to specific hosts. In other words, your server will request an IP address from your local DHCP server, and your DHCP server is instructed to give a specific address to your server each time it asks for one. This is the method I prefer, because it makes your DHCP server the single source of truth for the IP addresses that are assigned on your network. I'll go over it in more detail in *Chapter 11, Setting Up Network Services*.

However, you don't always have a choice. As a Linux administrator, you may or may not be in charge of the DHCP server. It's often the case at many organizations that administrators that manage servers are not the same individuals that also manage the network.

If you don't have authority over the design of the network, you'd use an IP address that would be provided to you by your network administrator, and then you'll proceed to configure your Ubuntu Server to use it by applying the parameters they give you.

Within the last several years, the method that we utilize to customize the IP address of our servers has changed from the way it was handled in the past. Since Ubuntu 17.10, which was released back in 2017, configuring your IP address settings is now done via Netplan. In the past, we would configure networking via NetworkManager, but that's installed only by default in Ubuntu Desktop. With Netplan, configuration files for your network interfaces now reside in the `/etc/netplan` directory, in YAML format. Explaining the YAML format itself is beyond the scope of this book, but the syntax is very easy to follow so you don't really need to thoroughly understand this format in order to configure your network interfaces. If you list the contents of the `/etc/netplan` directory, you should see at least one file there, often named `00-installer-config.yaml` or `50-cloud-init.yaml`. It's possible the file could be saved with a different name, so check the contents of the `/etc/netplan` directory to see what the file is named on your end. On one of my servers, I see the following contents in the `/etc/netplan/00-installer-config.yaml` file:

```
# This is the network config written by 'subiquity'
network:
  ethernets:
    enp0s3:
      dhcp4: true
  version: 2
```

We can already glean some obvious information from this default file. First, the comment at the beginning mentions `subiquity`, which is the official name of the Ubuntu Server installer, used when you install the distribution from boot media created from the ISO file.

More importantly, we can see that this particular server is configured to utilize DHCP in order to grab an IP address, which we can tell from the following line:

```
dhcp4: true
```

We can also tell that this configuration file pertains to interface `enp0s3`. Putting it all together, this file is telling us that interface `enp0s3` is configured to automatically obtain an IP address via DHCP. If we'd like to convert this configuration to a static IP address instead, we should first back up the file:

```
sudo cp /etc/netplan/00-installer-config.yaml /etc/netplan/00-installer-config.yaml.bak
```

This way, if we make a mistake, we can easily restore the original file by renaming the backup file to the original name. This is a good practice to get into, regardless of the file we're editing. Being able to restore a previous configuration is a best practice for just about any change we might be making. The first change we need to make is to remove the following line (or just change it to `false`):

```
dhcp4: true
```

Essentially, to set up a static IP we will replace that line with the details specific to our configuration. Here's an example of the file, configured for a static IP address:

```
# This is the network config written by 'subiquity'
network:
  ethernets:
    enp0s3:
      addresses: [192.168.100.50/24]
      gateway4: 192.168.100.1
      nameservers:
        addresses: [192.168.100.1, 192.168.100.2]
  version: 2
```

In the example, I've bolded four lines, which were added in place of the `dhcp4: true` line. First, we set the actual IP address:

```
addresses: [192.168.100.50/24]
```

There, I've used an example IP address of `192.168.100.50/24`. On your end, you would make sure that the IP address you choose is within the scope of the network you'd like your server to be a part of. As I mentioned earlier, the IP address you choose should *not* be within the DHCP scope that automatically assigned IP addresses are chosen from. The preceding IP address would be fine if the example scope of the DHCP server on your network ranged from `192.168.100.100` to `192.168.100.150`. The IP chosen here of `192.168.100.50` is outside of that, so we don't have to worry about another device being assigned that address.

We also include `/24` to declare that the IP address is part of a 24-bit **subnet**, which is fairly standard unless your network administrator has set up a larger scope. An `/24` network is the same as a Class C network, in case that's more familiar to you. This also takes care of the subnet mask, which we don't need here since `/24` implies the same subnet mask as `255.255.255.0` (if you're more familiar with the classful style, which shows the subnet mask in the same format as an IP address). We will discuss subnets in *Chapter 11, Setting Up Network Services*; however, a full walk-through of subnetting and the TCP/IP protocol would be more fitting for a book dedicated to networking concepts, so we won't go anymore into detail than necessary.

Moving on, we also set up the gateway:

```
gateway4: 192.168.100.1
```

When it comes to networking, the gateway refers to the device your outbound connections are routed through, which will often be a router or firewall, depending on how your network is set up. This value will need to match the actual default gateway address on your network, which is quite often the same as the IP address with the last portion being `.1`. If in doubt, you can check the IP address assignment of another device on the same network you're joining the Ubuntu server to, which would be the same.

The last section allows us to configure the DNS server that our server will use to look up external domain names:

```
nameservers:  
  addresses: [192.168.100.1, 192.168.100.2]
```

The example configuration is just that, an example—all the values must match whatever is appropriate for your network. Often, the DNS server IP address will be the same as the gateway address, but that's not always true. Sometimes a network administrator will have a custom IP scheme for DNS servers. I also added a secondary DNS server in the example, 192.168.100.2, but you can remove the second IP address if that's not necessary.

Once you've ensured that the values in the file are appropriate, we'll need to apply and test these changes:

- If you are using a virtual machine, you may want to make the changes from the virtual machine console
- If you're updating a physical machine, you may want to have a display and keyboard attached
- Although we discuss OpenSSH later in this chapter, if you're already aware of how to connect to a server via OpenSSH, you probably won't want to change network configuration over OpenSSH, since as soon as you activate these changes your connection will drop

Just take your time and double-check everything to ensure you didn't mistype anything, so you won't find yourself with a server that cannot connect.

To actually make these changes take effect, you can run the following command:

```
sudo netplan apply
```

When you run the previous command, it will let you know if there are any errors in the file or apply the changes if not. The new IP address will take effect immediately.

In the case of utilizing remote connections such as OpenSSH while configuring networking, you can work around the issue of being disconnected and having networking not restart properly by using `tmux`, a popular terminal multiplexer. A full run-through of `tmux` is beyond the scope of this book, but it is helpful to us in this scenario because it keeps commands running in the background, even if our connection to the server gets dropped. To use it, first install the package:

```
sudo apt install tmux
```

Then, activate `tmux` by simply typing `tmux` in your shell prompt.

From this point on, `tmux` is now responsible for your session. If you run a command within `tmux`, it will continue to run, regardless of whether or not you're attached to it. To see this in action, first enter `tmux` and then execute the `top` command. While `top` is running, disconnect from `tmux`. To do that, press `Ctrl + b` on your keyboard, release, and then press `d`. You'll exit `tmux`, but if you enter the `tmux` a command to reattach your session, you'll see that `top` was still running even though you disconnected. Following this same logic, you can activate `tmux` prior to executing the `sudo netplan apply` command.

Most likely, you'll still get dropped from your shell, since the process of activating network changes brings the network interface down and then back up again, but with `tmux` the command will complete in the background. You can then reconnect to the server and run `tmux a` to rejoin your `tmux` session.

The `tmux` utility is extremely powerful, and when harnessed can really enhance your workflow when using the Linux shell. Although a complete tutorial is outside the scope of this book, I highly recommend looking into using it, which you can do here: <https://www.packtpub.com/hardware-and-creative/getting-started-tmux>. If you need some guidance with it, check out the video guide on LearnLinuxTV that will be linked at the end of this chapter.

With networking restarted, you should be able to immediately reconnect to the server and see that the new IP assignment has taken place by executing `ip a`. If, for some reason, you cannot reconnect to the server, you may have made a mistake while editing the configuration file for Netplan. Double-check that file for any errors. But as long as you've followed along and typed in the proper values for your interface and network, you should be up and running with a static IP assignment.

Now, we have an actual network—we've named our server(s) and configured our network interfaces. We should also understand how name resolution works in Ubuntu, which is the process in which servers are able to find other servers by their name.

Understanding Linux name resolution

In *Chapter 11, Setting Up Network Services*, we'll have a discussion on setting up a DNS server for local name resolution for your network. But before we get to that, it's also important to understand how Linux resolves names in the first place. Most of you are probably aware of the concept of a **Domain Name System** (DNS), which matches human-understandable domain names to IP addresses. This makes browsing your network (as well as the internet) much easier. However, a DNS isn't always the first thing that your Linux server will use when resolving names.

For more information on the order in which Ubuntu Server checks resources to resolve names, feel free to take a look at the `/etc/nsswitch.conf` file. There's a line in this file that begins with the word `hosts`. Here is the output of the relevant line from the file on my server:

```
hosts:          files mdns4_minimal [NOTFOUND=return] dns mymachines
```

In this case, the server is configured to first check local files, and then the DNS if the request isn't found. This is the default order, and I see little reason to make any changes here (but you certainly can). Specifically, the file the server will check is `/etc/hosts`. If it doesn't find what it needs there, it will move on to the DNS (basically, it will check the DNS server we configured earlier with Netplan, or the default server provided by DHCP).

There are many other lines in the `nsswitch.conf` file, but I won't discuss them here as they are out of scope of the topic of this section.

The `/etc/hosts` file, which we briefly discussed while working with our `hostname`, tells our server how to resolve itself (it has a `hostname` mapping to the `localhost` IP of `127.0.0.1`), but you are also able to create additional names to IP mappings here as well.

For example, if I had a server (`myserver.mydomain.org`) at IP `10.10.96.124`, I could add the following line to `/etc/hosts` to make my machine resolve the server to that IP each time, without it needing to consult a DNS server at all:

```
10.10.96.124 myserver.mydomain.org
```

In practice though, this is usually not a very convenient method by which to configure name resolution. Don't get me wrong, you can certainly list your servers in this file along with their IP addresses, and your server would be able to resolve those names just fine. The problem stems from the fact that this method is difficult to maintain. The name mappings apply only to the server you've made the `/etc/hosts` changes on; other servers wouldn't benefit since they would only check their own `/etc/hosts` file. You could add a list of servers to the `hosts` file on each of your servers, but that would be a pain to manage. This is the main reason why having a central DNS server is a benefit to any network, especially for resolving the names of local resources.

However, the `/etc/hosts` file is used every now and again in the enterprise as a quick one-off workaround, and you'll probably eventually end up needing to use this method for one reason or another. One very common reason to use such a manual method of resolving names is in the case where you're testing a replacement server. In that case, you can configure the `/etc/hosts` file to have the same name as the original server, but with the IP address of the new server. Once you finish testing and confirm that the new server is operating properly, you can then replace the network-wide DNS name to point to the new IP address.

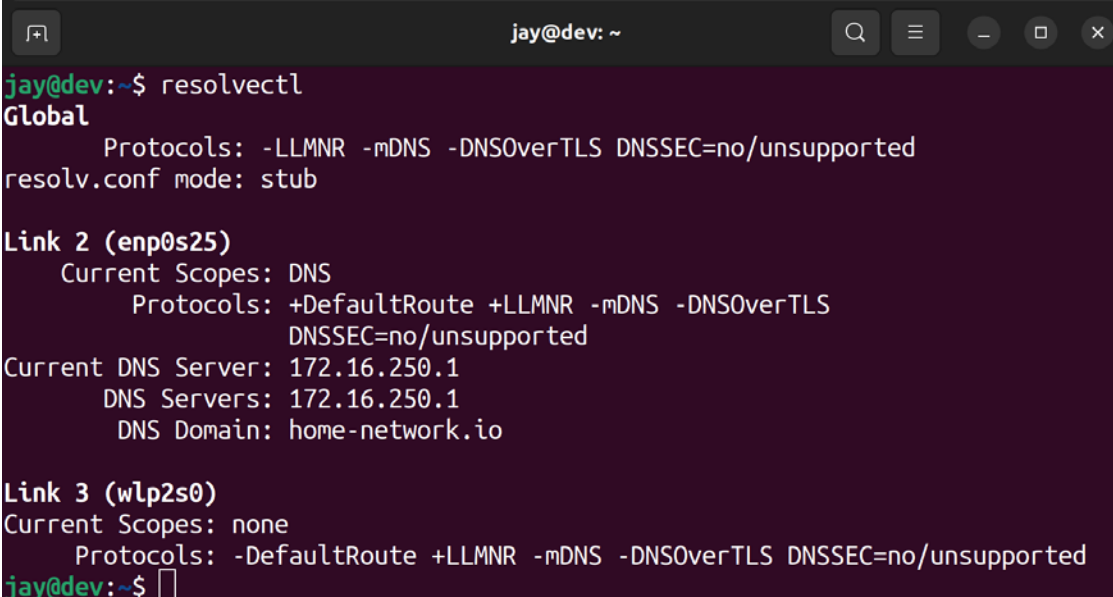
On legacy Ubuntu servers, the `/etc/resolv.conf` file included the IP addresses for DNS servers the system would use to resolve names. If you wanted to override the DNS servers for your server, you would alter that file. Although this file still exists in Ubuntu 22.04, it only exists to redirect lookups to `systemd-resolved`, which is a `systemd` unit that runs in the background and applies DNS settings based on what the system receives via DHCP or what you may have configured in Netplan. For the sake of completeness though, here is a brief overview of the syntax of this file in older releases, in case you end up working on such a server. An example of this file is as follows:

```
nameserver 10.10.96.1
nameserver 10.10.96.2
```

In this example, the `/etc/resolv.conf` output is utilizing servers `10.10.96.1` and `10.10.96.2`. Therefore, the server will first check `/etc/hosts` for a match of the resource you're looking up, and if it doesn't find it, it will then check `/etc/resolv.conf` in order to find out which server to check next. In this case, the server will check `10.10.96.1`.

The `/etc/resolv.conf` file on legacy servers is typically not one that you'd make actual changes to, since it's automatically generated by NetworkManager. NetworkManager is a service that helps you manage your network interfaces; however, it's not used anymore with Ubuntu Server for a handful of releases now. Even though you don't typically manually edit the `/etc/resolv.conf` file, it may be worth looking at on legacy servers to see which DNS servers were assigned, in case you're troubleshooting some sort of networking issue.

Nowadays, modern Ubuntu servers utilize `systemd-resolved` to handle name resolution. If you'd like to see which DNS servers were assigned on a newer Ubuntu Server installation, you can simply look at the configuration file for Netplan that we worked through earlier in the case of a static IP assignment, but if DHCP is being used, the `resolvectl` command will let you know what DNS nameservers your server is currently pointing to. This will provide an output similar to that shown in the following screenshot:



```
jay@dev: ~$ resolvectl
Global
  Protocols: -LLMNR -mDNS -DNSOverTLS DNSSEC=no/unsupported
resolv.conf mode: stub

Link 2 (enp0s25)
  Current Scopes: DNS
  Protocols: +DefaultRoute +LLMNR -mDNS -DNSOverTLS
             DNSSEC=no/unsupported
Current DNS Server: 172.16.250.1
DNS Servers: 172.16.250.1
DNS Domain: home-network.io

Link 3 (wlp2s0)
Current Scopes: none
Protocols: -DefaultRoute +LLMNR -mDNS -DNSOverTLS DNSSEC=no/unsupported
jay@dev: ~$
```

Figure 10.5: Viewing a server's current DNS assignment

In a typical enterprise Linux network, you'll set up a local DNS server to resolve your internal resources, which will then forward requests to a public DNS server in case you're attempting to reach something that's not internal. We'll get to that in *Chapter 11, Setting Up Network Services*, but you should now understand how the name resolution process works on your Ubuntu Server.

As Linux administrators, we will likely manage a large number of servers, and often the server we're managing may not even be in the same physical location as us. OpenSSH is a powerful tool for remote management, and it's what we'll explore next.

Getting started with OpenSSH

OpenSSH is quite possibly the most useful tool in existence for managing Linux servers. Of all the countless utilities available, this is the one I recommend that everyone starts practicing as soon as they can. Technically, I could probably better fit a section for setting up OpenSSH in *Chapter 11, Setting Up Network Services*, but this utility is very handy, and we should start using it as soon as possible.

OpenSSH allows you to open a command shell on other Linux servers, enabling you to run commands as if you were there in front of the server. For Linux administrators like us, this is extremely convenient. We could be tasked with managing dozens, hundreds, or even thousands of servers.

With OpenSSH, we can manage our entire server architecture without even getting out of our chairs. In this section, I'll give you some information on OpenSSH and how to install it, and then I'll finish up the section with a few examples of actually using it.

Installing OpenSSH

OpenSSH consists of two components, the server daemon that runs in the background that accepts SSH connections, and the client that runs on a laptop, workstation, or another server that gives you the ability to connect to an SSH server and run commands. All operating systems nowadays give you access to an OpenSSH client that you can use to make the connection to the server, so that requirement is probably already met. When it comes to Linux, most distributions give you the OpenSSH client already. You can verify that by running `which ssh` at your shell prompt. If you have the client installed, your output should read `/usr/bin/ssh`.

If, for some reason, you don't have this package installed and you've received no output from the previous command (which would be rare), you can install the OpenSSH client with the following command:

```
sudo apt install openssh-client
```

Depending on the choices you've made during installation, your Ubuntu Server likely has the OpenSSH server installed already. If you don't remember whether or not you opted to have this included during our initial installation, you can run the `which sshd` command at your shell prompt and you should see the output of `/usr/sbin/sshd`. You can also execute `systemctl status ssh` as well, and if the server component is present and running, then your server is ready to accept SSH connections:



```
jay@dev: ~  
jay@dev:~$ systemctl status ssh  
● ssh.service - OpenBSD Secure Shell server  
   Loaded: loaded (/lib/systemd/system/ssh.service; enabled; vendor preset: enabled)  
   Active: active (running) since Mon 2022-05-09 00:15:11 UTC; 1min 30s ago  
     Docs: man:sshd(8)  
           man:sshd_config(5)  
  Main PID: 9403 (sshd)  
    Tasks: 1 (limit: 9251)  
   Memory: 1.7M  
      CPU: 22ms  
   CGroup: /system.slice/ssh.service  
           └─9403 "sshd: /usr/sbin/sshd -D [listener] 0 of 10-100 startups"  
jay@dev:~$
```

Figure 10.6: Verifying that the OpenSSH service is running on a server

In the case that you don't have the OpenSSH server component installed on your server, you can install it with the following command:

```
sudo apt install openssh-server
```

With great power comes great responsibility, though. As great as OpenSSH is, any service that listens for connections has the potential of being abused. An outside intruder finding a weakness or exploit that allows them to take control over your server is a really bad situation. Therefore, like any service that runs on your server, you should only have it running if you need to use it. Since OpenSSH is incredibly helpful (and it's the standard method of remote management) it's very hard to *not* use it.

There are many methods you can utilize to secure such a service and help protect it. One method is covered in the next section, and we'll talk about security pertaining to OpenSSH again before the book comes to a close. Specifically, in *Chapter 21, Securing Your Server*, I will walk you through various configuration changes you can make to help minimize the threat of miscreants breaking into your server from the outside and wreaking havoc.

Securing OpenSSH is actually not hard at all, and would probably only take a few minutes of your time. Therefore, feel free to make a detour to that chapter to read the section there that talks about securing OpenSSH, and then come back here when you're done. For now, make sure that you have secure, randomly generated passwords on the server at the very least. If OpenSSH is reachable via the public internet, and any of your users have weak passwords, it definitely won't be a fun situation.

With all of that out of the way, we can get started with actually using OpenSSH.

Issuing commands with OpenSSH

After you've installed the `openssh-server` package on your target machine (the one you want to control remotely), you'll need to start it if it hasn't been already. By default, Ubuntu's `openssh-server` package is automatically configured to start and become enabled once installed. Just as we've done earlier, you can verify that the required service is running with the following command:

```
systemctl status ssh
```

If OpenSSH is running as a daemon on your server, you should see output that tells you that it's active (running). If not, you can start it with the following command:

```
sudo systemctl start ssh
```

If the output of the `systemctl status ssh` command shows that the daemon is disabled (meaning it doesn't start up automatically when the server boots), you can enable it with the following command:

```
sudo systemctl enable ssh
```

With the OpenSSH server started and running, your server should now be listening for connections. To verify this, use the following command to list listening ports, restricting the output to SSH:

```
sudo ss -tunlp |grep ssh
```

The `ss` command allows us to view a list of processes running on our server that are listening for connections. It will also display which port a process is listening on.

We'll explore this command in more detail in *Chapter 21, Securing Your Server*. But for now, this command should produce an output similar to the following:

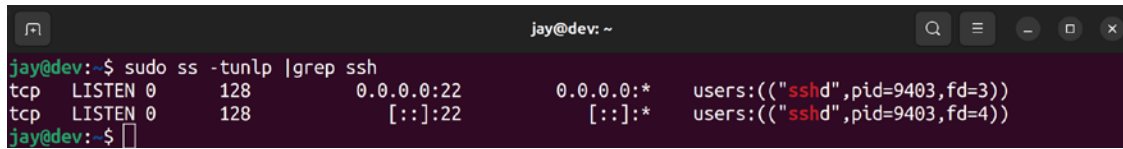
A terminal window with a dark purple background. The prompt is 'jay@dev: ~'. The command entered is 'sudo ss -tunlp |grep ssh'. The output shows two lines of text: 'tcp LISTEN 0 128 0.0.0.0:22 0.0.0.0:* users:(("sshd",pid=9403,fd=3))' and 'tcp LISTEN 0 128 [::]:22 [::]:* users:(("sshd",pid=9403,fd=4))'. The prompt returns to 'jay@dev:~\$'.

Figure 10.7: Checking if the required port for SSH is listening

If, for some reason, your server doesn't show that it has an SSH server listening, double-check that you've started the daemon. By default, the SSH server listens for connections on port 22. This can be changed by modifying the port declaration in the `/etc/ssh/sshd_config` file, but that's a story for a later chapter. While I won't be going over the editing of this file just yet, just take note that it's the default configuration file for the daemon. OpenSSH reads this file for configuration values each time it's started or restarted.

To connect to a server using SSH, simply execute the `ssh` command followed by the name or IP address of the server you'd like to connect to:

```
ssh 192.168.1.120
```

By default, the `ssh` command will use the username you're currently logged in with for the connection. If you'd like to use a different username, specify it with the `ssh` command by including your username followed by the `@` symbol just before the IP address or hostname:

```
ssh fmulder@192.168.1.120
```

Unless you tell it otherwise, the `ssh` command assumes that your target is listening on port 22. If it isn't, you can give the command a different port with the `-p` option followed by a port number:

```
ssh -p 2242 fmulder@192.168.1.120
```

Once you're connected to the target machine, you'll be able to run shell commands and administer the system as if you were right in front of it. You'll have all the same permissions as the user you've logged in with, and you'll also be able to use `sudo` to run administrative commands if you normally have access to do so on that server.

Basically, anything you're able to do if you were standing right in front of the server, you'll be able to do via SSH. When you're finished with your session, simply type `exit` at the shell prompt, or press `Ctrl + d` on your keyboard.

When you exit an OpenSSH connection, any processes you may have had running in the background will be killed. Be sure you resume any background processes you may have running and finish working with them before you exit your connection. We took a look at how to run processes in the background back in *Chapter 7, Controlling and Monitoring Processes*.

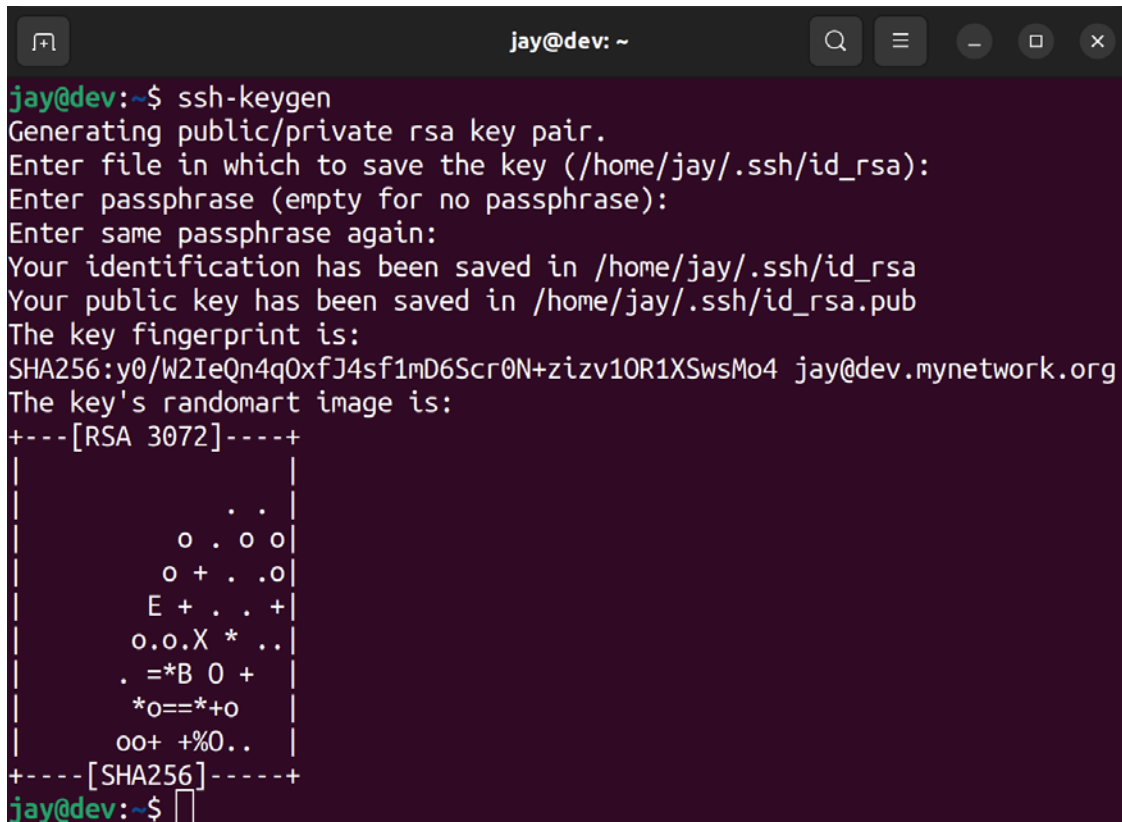
As you can see, OpenSSH is a miraculous tool that will benefit you by allowing you to remotely manage your servers from anywhere you allow SSH access from. Make sure to read the relevant section in *Chapter 11, Setting Up Network Services*, with regard to securing it, though. In the next section, we'll discuss SSH key management, which brings the benefit of convenience, but also allows you to increase security as well.

Getting started with SSH key management

When you connect to a host via SSH, you'll be asked for your password, and after you authenticate you'll be connected. Instead of using your password, though, you can authenticate via public key authentication instead. The core benefit to this method is added security, as your system password is never transmitted during the process of connecting to the server. When you create an SSH key pair, you are generating two files, a *public key* and a *private key*. These two files are mathematically linked, so if you connect to a server that has your public key, it will know it's you because you (and only you) have the private key that matches it. This method is far more secure than password authentication, and I highly recommend that you use it. To get the most out of the security benefit of authentication via keys, you can actually disable password-based authentication on your server so that your SSH key is your only way in. By disabling password-based authentication and using only keys, you're increasing your server's security by a sizeable margin. We'll go over that in *Chapter 21, Securing Your Server*.

Generating public and private keys

To get started, you'll first need to generate your key. To do so, on your workstation or laptop (the device you're using to connect to the server), use the `ssh-keygen` command as your normal user account. The following screenshot shows what this process generally looks like:



```

jay@dev: ~
jay@dev:~$ ssh-keygen
Generating public/private rsa key pair.
Enter file in which to save the key (/home/jay/.ssh/id_rsa):
Enter passphrase (empty for no passphrase):
Enter same passphrase again:
Your identification has been saved in /home/jay/.ssh/id_rsa
Your public key has been saved in /home/jay/.ssh/id_rsa.pub
The key fingerprint is:
SHA256:y0/W2IeQn4q0xfJ4sf1mD6Scr0N+zizv10R1XSwsMo4 jay@dev.mynetwork.org
The key's randomart image is:
+---[RSA 3072]-----+
|
|      . .
|     o . o o
|    o + . .o
|   E + . . +
|  o,o,X * ..
| . =*B O +
| *o==*+o
| oo+ +%O..
+-----[SHA256]-----+
jay@dev:~$ █

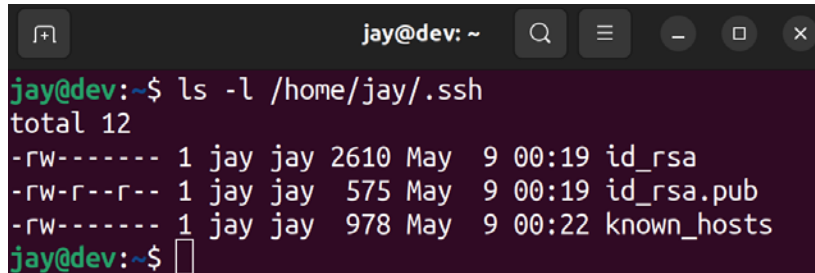
```

Figure 10.8: Generating an SSH key pair

First, you'll be asked for the directory in which to save your key files, defaulting to `/home/<user>/.ssh`. You'll next be asked for a passphrase, which is optional. Although it does add an additional step to authenticating via keys, I recommend that you give it a passphrase (which should be different than your system password) since it greatly enhances security (the key would be unusable without the passphrase, if you gave it one). You can press *Enter* for the passphrase without entering one if you do not want this.

What this command does is create a directory named `.ssh` in your home directory, if it doesn't already exist. Inside that directory, it will create two files, `id_rsa` and `id_rsa.pub`. The `id_rsa` file is your private key. It should never leave your machine, be given to another user, or be stored on any external media. If your private key leaks out, your keys can no longer be trusted. By default, the private key is owned by the user that created it, with `rw` permissions given only to its owner.

The public key, on the other hand, can leave your computer and doesn't need to be secured as much. Its permissions are more lenient, being readable by everyone and writable by the owner. You can see this yourself by executing `ls -l /home/<user>/.ssh`:



```

jay@dev: ~
jay@dev:~$ ls -l /home/jay/.ssh
total 12
-rw----- 1 jay jay 2610 May  9 00:19 id_rsa
-rw-r--r-- 1 jay jay  575 May  9 00:19 id_rsa.pub
-rw----- 1 jay jay  978 May  9 00:22 known_hosts
jay@dev:~$ █

```

Figure 10.9: Listing the contents of the `.ssh` directory, showing the permissions of the newly created keys

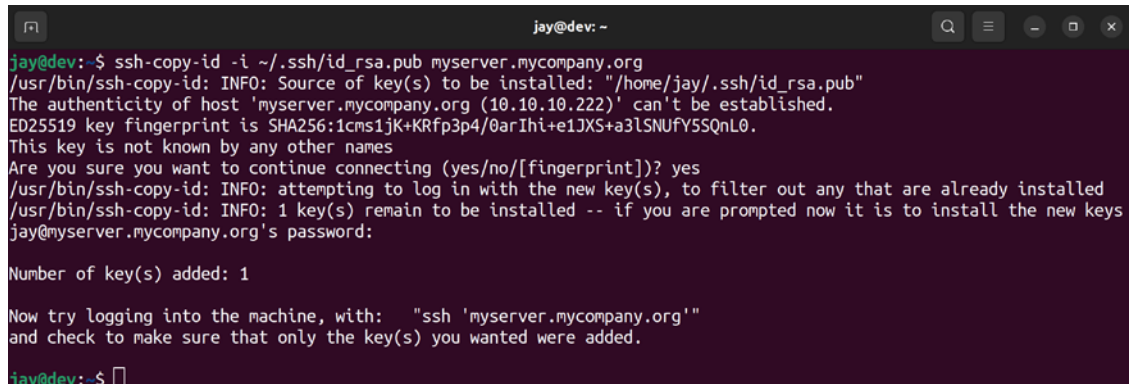
The public key is the key that actually gets copied to other servers to facilitate your being able to log in via such a key pair. When you log in to a server that has your key, it checks that it's a mathematical match to your private key, and if it is, it will let you log in. You'll also be asked for your passphrase, if you chose to set one when you first created it. But before we can actually use our key, we need to copy it over to the server we intend to connect to.

Copying your public key to a remote server

To actually transmit your public key to a target server, we can use the `ssh-copy-id` command, as I'm doing in the following example command:

```
ssh-copy-id -i ~/.ssh/id_rsa.pub 192.168.1.150
```

With that command, replace the IP address with the actual IP address, or with the *hostname* of the target server. You'll be asked to log in via your password first, and then your key will be copied over. From that point on, you'll log in via your key, falling back to being asked for your password if, for some reason, your key relationship is broken. Here's an example of what this process looks like, if I were to work through the process of copying my key to a server named `myserver.mycompany.org`:



```

jay@dev: ~
jay@dev:~$ ssh-copy-id -i ~/.ssh/id_rsa.pub myservers.mycompany.org
/usr/bin/ssh-copy-id: INFO: Source of key(s) to be installed: "/home/jay/.ssh/id_rsa.pub"
The authenticity of host 'myservers.mycompany.org (10.10.10.222)' can't be established.
ED25519 key fingerprint is SHA256:1cms1jK+KRfp3p4/0arIhi+e1JXS+a3LSNUFY5SQnL0.
This key is not known by any other names
Are you sure you want to continue connecting (yes/no/[fingerprint])? yes
/usr/bin/ssh-copy-id: INFO: attempting to log in with the new key(s), to filter out any that are already installed
/usr/bin/ssh-copy-id: INFO: 1 key(s) remain to be installed -- if you are prompted now it is to install the new keys
jay@myservers.mycompany.org's password:

Number of key(s) added: 1

Now try logging into the machine, with: "ssh 'myservers.mycompany.org'"
and check to make sure that only the key(s) you wanted were added.
jay@dev:~$ █

```

Figure 10.10: Using the `ssh-copy-id` command to copy a public key to a server

So, what exactly did the `ssh-copy-id` command do? Where is your public key copied to, exactly? What happens with this command is that on the target server, a `.ssh` directory is created in your home directory on the target server if it didn't already exist. Inside that directory, a file named `authorized_keys` is created if it wasn't already present. The contents of `~/.ssh/id_rsa.pub` on your machine are copied into the `~/.ssh/authorized_keys` file on the target server. With each additional key you add (for example, you connect to that server from multiple machines), the key is added to the end of the `authorized_keys` file, one per line.

Using the `ssh-copy-id` command is merely a matter of convenience; there's nothing stopping you from copying the contents of your `id_rsa.pub` file and manually pasting it into the `authorized_keys` file of the target server. That method will actually work just fine as well.

When you connect to a server that you have set up a key relationship with by adding your public key, SSH checks the contents of the `~/.ssh/authorized_keys` file on that server, looking for a key that mathematically matches the private key (`~/.ssh/id_rsa`) on your machine. If the two keys are an appropriate match, you are allowed access. If you set up a passphrase, you'll be asked to enter it in order to open your public key.

If you decided not to create a passphrase with your key, you're essentially setting up authentication without a password, meaning you won't be asked to enter anything when authenticating.

Utilizing an SSH agent

When we created our SSH key earlier, it was mentioned that having a passphrase is optional but is a good idea. Using passphrases with OpenSSH key pairs boosts their security. If an OpenSSH key falls into the wrong hands, it will be useless if the person attempting to utilize it doesn't know the passphrase. However, we lose a bit of convenience because we need to enter the passphrase for a key each time we want to use it. An OpenSSH key without a passphrase will allow us to connect to a server and be logged in without entering anything at all. With an *SSH agent*, you can actually cache your passphrase the first time you use it, so you won't be asked for it with every connection. This essentially allows you to benefit from the added security of a passphrase, and still maintain at least some convenience. Best of all, if your laptop or desktop is able to utilize the OpenSSH client for connecting to remote systems, you should have an SSH agent on your system already. If we're using a flavor of Linux or macOS on our workstation or laptop, for example, we will have the `ssh-agent` command available to us.

The `ssh-agent` is used by starting it in the background in our terminal. We can then “unlock” our keys with our passphrase, and then the unlocked key will be stored in memory and will be automatically used when we attempt to connect to a server we've copied our public key to. To start it, enter the following command as your normal user account on the machine you're starting your connections from (that is, your workstation):

```
eval $(ssh-agent)
```

This command will start an SSH agent, which will continue to run in the background of your shell. But it's not adding any value to us yet—so we will need to add an SSH key to the now-running agent. The `ssh-add` command allows us to add an SSH key to our running `ssh-agent`. To do so, we can give the `ssh-add` command the path to our public key as an argument:

```
ssh-add ~/.ssh/id_rsa
```

At this point, you'll be asked for your passphrase. As long as you enter it properly, your key will remain open and you won't need to enter it again for future connections, until you close that shell or log out. Now that you have the `ssh-agent` running in the background with your unlocked key, utilizing a key with a passphrase becomes much easier and you'll end up typing a lot less.

Changing the passphrase of an OpenSSH key

At some point, you may want to change the passphrase associated with a key. If you'd like to do that, you can use the `-p` argument with the `ssh-keygen` command. That same argument can also be used if you didn't choose to add a passphrase when you initially created the key. That's all there is to it. The command you'd enter to add or change a passphrase is as simple as the following:

```
ssh-keygen -p
```

Once you enter that command, press *Enter* to accept the default file (`id_rsa`) unless the key you want to alter is under a different name, in which case you can type the path to, and the name of, that key. Then, you'll be asked for your current passphrase (leave it blank if you don't have one yet) followed by your new passphrase twice. The process looks like this:

A terminal window titled 'jay@dev: ~' showing the execution of the 'ssh-keygen -p' command. The output prompts the user to enter the file path (defaulting to '/home/jay/.ssh/id_rsa'), shows the key's comment as 'jay@dev.mynetwork.org', and asks for the current and new passphrases. The process concludes with the message 'Your identification has been saved with the new passphrase.' and the prompt returns to 'jay@dev: ~\$' with a cursor.

```
jay@dev: ~$ ssh-keygen -p
Enter file in which the key is (/home/jay/.ssh/id_rsa):
Key has comment 'jay@dev.mynetwork.org'
Enter new passphrase (empty for no passphrase):
Enter same passphrase again:
Your identification has been saved with the new passphrase.
jay@dev: ~$
```

Figure 10.11: Changing an SSH passphrase

These concepts may take a bit of practice if you've never used SSH before. The best way to practice is to set up multiple Ubuntu Server installations (perhaps several virtual machines), and practice using SSH to connect to them, as well as deploying your key to each machine via the `ssh-copy-id` command. It's actually quite easy once you get the hang of it.

Simplifying SSH connections with a config file

Before we leave the topic of OpenSSH, there's another trick that has the benefit of convenience, and that is the creation of a local configuration file for SSH. This file must be stored in the `.ssh` directory of your home directory, and be named `config`. The full path for this file in my case looks like this:

```
/home/jay/.ssh/config
```

This file doesn't exist by default, but if it's found, SSH will parse it whenever you use the client and you'll be able to benefit from it. Go ahead and open this file in your text editor, such as nano:

```
nano /home/your_username/.ssh/config
```

This config file allows you to type configuration for servers that you connect to often, which can simplify the ssh command automatically. The following are example contents from such a file that will help me illustrate what it does:

```
host myserver
  Hostname 192.168.1.23
  Port 22
  User jdoe

Host nagios
  Hostname nagios.mynetwork.org
  Port 2222
  User nagiosuser
```

In the example contents, I have two hosts outlined, myserver and nagios. For each, I've identified a way to reach it by name or IP address (the Hostname line), as well as the Port and User account to use for the connection. If I use ssh to connect to either Host by the name I outlined in this file, it will use the values I have stored there, for example:

```
ssh nagios
```

That command is a lot shorter than if I set all the options manually. Considering I have a config file for SSH, that command is essentially the same as if I identified the connection details manually, which would've been the following:

```
ssh -p 2222 nagiosuser@nagios.mynetwork.org
```

I'm sure you can see how much simpler it is to type the first command than the second. With the config file for SSH, I can have some of those details automatically applied. Since I've outlined that my nagios server is located at nagios.mynetwork.org, its SSH user is nagiosuser, and it's listening on port 2222, it will automatically use those values even though I only typed ssh nagios. Furthermore, you can also override this entry as well. If you provide a different username when you use the ssh command, it will use that instead of what you have written in the config file.

In the first example (for the myserver server), I'm providing an IP address for the connection, rather than a hostname. This is useful in a situation where you may not have a DNS entry for your target server. With this example, I don't have to remember that the IP address for myserver is 192.168.1.23. I simply execute ssh myserver and it's taken care of for me.

The names of each server within the `config` file are arbitrary and don't have to match the target server's hostname. I could've named the first server `potato` and it would still have routed me to `192.168.1.23`, so I can create any sort of named shortcut I want, whatever I find is most convenient for me and easiest to remember. As you can see, maintaining a `config` file in your home directory containing your most commonly used SSH connections will certainly help keep you organized and allow you to connect more easily.

Summary

In this chapter, we worked through several examples of connecting to other networks. We started off by configuring our hostname, managing network interfaces, assigning static IP addresses, as well as looking at how name resolution works in Linux. A decent portion of this chapter was dedicated to topics regarding OpenSSH, which is an extremely useful utility that allows you to remotely manage your servers. We'll revisit OpenSSH in *Chapter 21, Securing Your Server*, with a look at boosting its security. Overall, we've only begun to scratch the surface of this tool. Entire books have been written about SSH, but the examples in this chapter should be enough to make you productive with it. The name of the game is to practice, practice, practice!

In the next chapter, we'll talk about managing software packages. We'll work through adding and removing them, adding additional repositories, and more!

Relevant videos

- OpenSSH Guide (LearnLinuxTV): <https://linux.video/ssh-guide>
- tmux Guide (LearnLinuxTV): <https://linux.video/tmux-guide>
- The OpenSSH Client Config File (LearnLinuxTV): <https://linux.video/ssh-config-file>
- Getting Started with OpenSSH Key Management (LearnLinuxTV): <https://linux.video/ssh-key-mgmt>

Further reading

- Netplan FAQ: <https://learnlinux.link/npf>
- Ubuntu SSH key documentation: <https://learnlinux.link/u-ssh-keys>
- NetworkInterfaceNames (from Debian's wiki): <https://learnlinux.link/net-int-nam>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



11

Setting Up Network Services

In *Chapter 10, Connecting to Networks*, we went over some important foundational topics related to networking. We saw how to set the hostname, manage network interfaces, configure connections, and more. In this chapter, we'll revisit networking, specifically to set up the resources that will serve as the foundation of our network. The majority of this chapter will focus on setting up the DHCP and DNS servers, which are very important components of any network. In addition, we'll even take a look at setting up a server to act as an internet gateway for the rest of our network.

Along the way, we'll cover the following topics:

- Planning your IP address scheme
- Serving IP addresses with `isc-dhcp-server`
- Setting up DNS with `bind`
- Setting up an internet gateway

As a Linux administrator, you may or may not be tasked with designing the entire network layout of your organization; often there will be an already-existing network to manage. In the next section, we'll discuss creating such a layout, even if only to understand what goes into such a plan.

Planning your IP address scheme

Designing the overall layout of your network is an incredibly important process that sets the stage for success or failure later on. This design must take into consideration the needs of the organization, the need for efficient methods of communication, and the segregation of network services to ensure that your servers can only communicate with the resources that they are supposed to. As a Linux administrator that manages a fleet of Ubuntu servers, it's not always the case that you'll even have a say in the network layout at all. It's quite common that you "inherit" a network designed by a previous administrator, or your job is siloed such that you only manage the servers and someone else is responsible for the network.

Since this is primarily a book that focuses on teaching you how to manage Ubuntu servers, we aren't going to cover all of the things that a network administrator would need to know, but there's a bit of overlap between the two roles. As a Linux administrator, you may or may not be tasked with the designing of your network, but at a minimum, you do need to understand the overall layout and how your servers will fit within it.

In this section, we'll discuss the most important part of a network layout—the IP address scheme. Planning the IP scheme is an important task that sets the foundation for many other things. Even if it's not up to you to design this layout, knowing the finer details of an existing layout can help you better understand how servers you maintain might communicate with other devices. Planning your IP layout involves estimating how many devices will need to connect to your network and being able to support them. In addition, a good plan will account for potential growth and allow expansion as well. The main thing that factors into this is the size of your user base. Perhaps you are working in a small office with only a handful of people, or a large corporation with thousands of users and hundreds of virtual machines. Even if your organization is only a small office, I always recommend making the assumption that your company will explode in popularity someday and designing your network to have the growth potential to accommodate that.

Typically, most off-the-shelf routers and network equipment come with an integrated **Dynamic Host Control Protocol (DHCP)** server, with a default class C (/24) network. Essentially, this means that if you do not perform any configuration at all, you're limited to 254 addresses. For a small office, this may seem like plenty. After all, if you don't even have 254 employees at your company, that number may seem like overkill. As I mentioned before, potential growth is always something to keep in mind. But even if we remove that from the equation, IP addresses are used up quicker than you'd think nowadays—even when it comes to internal addressing. Average users may consume three IP addresses each, and sometimes more.

For example, perhaps a user not only has a laptop (which itself can have both a wired and wireless interface, both consuming an IP address), but perhaps they also have a mobile phone (which likely features Wi-Fi), and a **Voice over IP (VoIP)** phone (there goes another address). If that user somehow manages to convince their supervisor that they also need a desktop computer as well as their laptop, there will be a total of five IP addresses for that one user. Suddenly, 254 addresses don't seem like all that many.

Perhaps a really good real-world example of the number of available IP addresses potentially being a problem is a small restaurant in a busy city that offers free Wi-Fi to its customers. The person designing the guest network for their customers may assume that a /24 network is more than enough if they only have one hundred customers or so each day. While that logic may seem sound, consider that if the restaurant is next to a busy street, people that have Wi-Fi enabled on their phone may snag an address as they pass by, and if the DHCP addresses are configured to expire in 24 hours, then there will often be no available IP addresses for actual customers most of the time. If you've ever attempted to connect to a restaurant Wi-Fi network and found yourself unable to access the internet after connecting, this could be a real-world example of a pain point of not designing a network layout effectively.

The obvious answer to this problem is splitting up your network into **subnets**. Although I won't go into the details of how to subnet your network (which would be beyond the scope of this book), I mention it here because it's definitely something you should take into consideration. In the next section, I'll explain how to set up your own DHCP server with a single network. However, if you need to expand your address space, you can easily do so by updating your DHCP configuration. When coming up with an IP address layout, always assume the worst and plan ahead. While it may be a simple task to expand your DHCP server, planning a new IP scheme rollout is very time-consuming, and to be honest, annoying.

When I set up a new network, I like to divide the address space into several categories. First, I'll usually set aside a group of IP addresses specifically for DHCP. These addresses will get assigned to clients as they connect, and I'll usually have them expire and need to be renewed in about one day. Then, I'll set aside a block of IP addresses for network appliances, another block for servers, and so on. In the case of a typical /24, I might decide on a scheme such as the following (assuming it's a small office with no growth planned):

```
Network: 192.168.1.0/24
Network equipment: 192.168.1.1 - 192.168.1.10
Servers: 192.168.1.11 - 192.168.1.99
DHCP: 192.168.1.100 - 192.168.1.240
Reservations: 192.168.1.241 - 192.168.1.254
```

Since I mentioned it's a good idea to plan for future growth, the /24 would be potentially constrained and wouldn't accommodate much growth. I chose that scheme to keep everything in this chapter simple for the sake of easy explanation. But in a real company network, you may want to consider a larger number of IP addresses than what the above would give you.

How do you get more IP addresses? Take a look at the number after the network address, which is /24 on the first line. With that number, we're configuring how large the IP address space is, which essentially correlates to how many IP addresses we have available. That number is known as a **CIDR** notation, which stands for **Classless Inter-Domain Routing**. Similar to a subnet mask, changing that number results in a different number of IP addresses. For example, if you change the /24 portion of the network to /22, you instantly have 1,022 possible addresses to work with, rather than 254. If you lower it again to /20, the number jumps to 4,094. For now, don't worry too much about this. You can focus on remembering that the higher the number, the fewer IP addresses you have. It can go up to /32, which only gives you one IP address. If you'd like to experiment further with subnetting, we've provided a subnet calculator in the *Further reading* section.

The following is a list of common CIDR values and their impact on the number of available IP addresses:

CIDR	Total Usable IP Addresses
/32	1
/24	254
/16	65,534
/8	16,777,214

Of course, no single IP address scheme is right for everyone. The scheme I provided earlier is simply a hypothetical example, so you shouldn't copy mine and use it on your network unless it matches your needs. I'll use this scheme for the remainder of this chapter, since it works fine as an example. To explain my sample rollout, we start off with a 24-bit network, `192.168.1.0/24`. The address `192.168.1.0` refers to the network itself, and that IP address is not assignable to clients. The first usable IP address in this subnet will be `192.168.1.1`. The last IP address in this block (`192.168.1.255`) is not assignable either, since that is known as the **broadcast address**. Anything that's sent to the broadcast address is effectively sent to every IP in the block, so we can't really use it for anything but broadcasts. In summary, keep in mind that an IP address ending in `.0` can't be used, and neither can an IP address that ends in `.255`.



Technically, you *can* assign IP addresses ending in `.0` or `.255`, depending on the CIDR value. However, since those ending values are most often used to refer to the network itself and the broadcast address respectively, some applications and pieces of hardware may not accept these values so for all intents and purposes, you should consider those ending values as unassignable.

Going back to the example IP layout mentioned earlier in this section, I set aside a group of IP addresses starting with `192.168.1.1` through `192.168.1.10` for use by network appliances. Typical devices that would fit into this category would be managed switches, routers, wireless access points, and so on. These devices typically have an integrated web console for remote management, so it would be best to have a static IP address assignment. That way, I'll have an IP address available that I can use to access these devices. I like to set up network appliances as the first devices so that they all get the lowest numbers when it comes to the last number of each IP address. This is just personal preference.

Next in the example layout, we define IP addresses `192.168.1.11` through `192.168.1.99` for servers. This may seem like quite a few addresses for servers, and it is. However, with the rise of virtualization and how simple it has become to spin up a server, this block could get used up faster than you'd think. Feel free to adjust accordingly.

Now we have our DHCP pool, which consists of addresses `192.168.1.101` through `192.168.1.240`. These IP addresses are assignable to any devices that connect to our network. Typically, I like to have these assignments expire in one day to prevent one-off devices from claiming and holding onto an IP address for too long, which can lead to devices fighting over a DHCP lease. In this situation, you'd have to clear your DHCP leases to reset everything, and I find that to be too much of a hassle. When we get to the section on setting up a DHCP server, I'll show you how to set the expiration time.

Finally, we have addresses `192.168.1.241` through `192.168.1.254` for the purposes of DHCP reservations. I generally refer to reserved DHCP addresses as *static leases*, but both terms mean the same thing. These addresses will be assigned by DHCP, but each device with a static lease will be given the same IP address each time. You don't have to separate these into their own pool, since DHCP will not assign the same address twice. It may still be a good idea to separate them though, if only to be able to tell from looking at an IP address that it's a static lease, due to it being within a particular hypothetical block. Static leases are good for devices that aren't necessarily a server but still need a predictable IP address.

An example of this may be an administrator's desktop PC. Perhaps they want to be able to connect to the office via VPN and be able to easily find their computer on the network and connect to it. If the IP was dynamically assigned instead of statically assigned, it would be harder for them to find it.

After you carve up your IP addresses, the next thing is to make sure that they're accurately documented. If you don't focus on documentation now while designing services, you will definitely regret it later. You can consider setting up a private wiki server, for example.

Another method is creating a spreadsheet to keep track of your static IP assignments. This is acceptable if you don't have a better solution; it doesn't have to be anything fancy. Among the usual components, such as the device info and IP address, I also include the MAC address of each device on the spreadsheet, which will come in handy when we set up our DHCP server in the next section:

Servers

Device	IP	MAC Address	Model
D-Link Access Point	192.168.1.2	E0:3B:49:6E:6C:BE	DAP-2695
Web Server	192.168.1.11	52:54:01:88:F8:BC	Dell Poweredge R710
Database Server	192.168.1.12	00:23:4D:A5:F2:EB	Dell Poweredge R610
Nagios	192.168.1.13	B8:27:EB:E2:29:03	Raspberry Pi 4
Bruce Wayne's Desktop	192.168.1.13	D0:51:99:37:A9:0D	System76 Thelio

Figure 11.1: An example IP address layout spreadsheet

As you can see from my example layout, our number of available addresses is rather limited with a 24-bit network. However, this layout will serve as an example we can follow that's good enough for the remainder of the chapter. At this point, just think about the factors that are important to your organization, and ensure that any networks you create are scalable and accommodate your needs.

Now that we have an IP layout (if we didn't have one already), we can take a look at setting up a DHCP server, which will be the service that will ultimately be assigning these IPs.

Setting up a DHCP server for serving IP addresses

Most network appliances you purchase nowadays often come with their own DHCP server and allow you to configure it via a web console. Often, this is totally fine and meets your needs. In my experience though, network appliances you purchase can be very hit and miss. Some of them are great, while others are not so impressive. One of the main problems is that manufacturers will often stop supporting the hardware prematurely, exposing your network to unpatched vulnerabilities. It's important to always purchase network hardware that's as future-proof as possible. Or, you can set up your own router with the features that you need. Although the burden of management is placed on you, this gives you ultimate flexibility. Ubuntu servers make great DHCP servers, and rolling your own server is actually easier than it sounds. And that's exactly what we're going to work through in this section.

First, the server that serves DHCP will definitely need a static IP address. This means you'll need to configure Netplan with a static IP assignment. A static lease won't work here, since the DHCP server can't assign an IP address to itself.

Also, the IP address that you designate for your server's static IP must be in the same network as the addresses that you plan to serve. Otherwise, the service will fail to start even after we configure it.

If you have yet to set a static IP address, *Chapter 10, Connecting to Networks*, has a section that will walk you through the process.

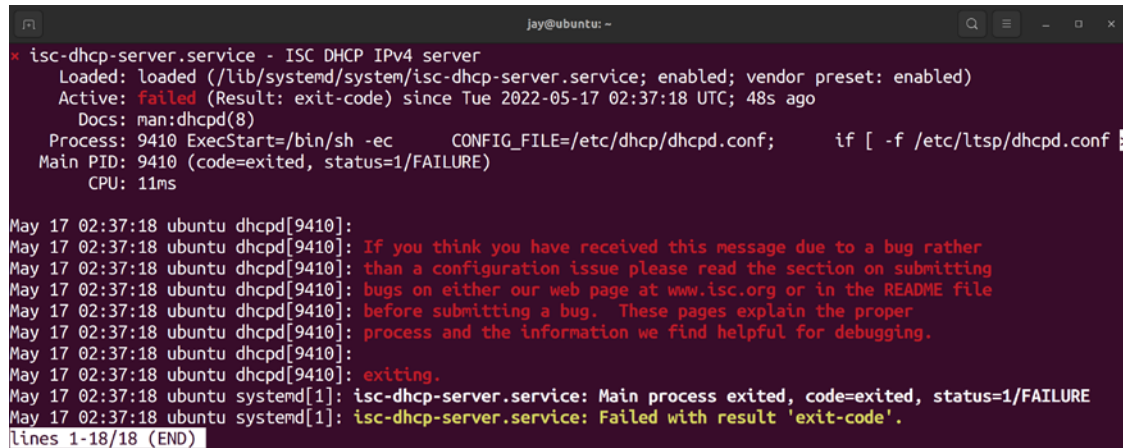
Once you assign a static IP address, the next step is to install the `isc-dhcp-server` package:

```
sudo apt install isc-dhcp-server
```

Check the status of the daemon after installing the `isc-dhcp-server` package, using the following command:

```
systemctl status isc-dhcp-server
```

You'll likely notice that it failed:



```

jay@ubuntu: ~
x isc-dhcp-server.service - ISC DHCP IPv4 server
   Loaded: loaded (/lib/systemd/system/isc-dhcp-server.service; enabled; vendor preset: enabled)
   Active: failed (Result: exit-code) since Tue 2022-05-17 02:37:18 UTC; 48s ago
     Docs: man:dhcpd(8)
   Process: 9410 ExecStart=/bin/sh -ec CONFIG_FILE=/etc/dhcp/dhcpd.conf; if [ -f /etc/ltsp/dhcpd.conf
   Main PID: 9410 (code=exited, status=1/FAILURE)
      CPU: 11ms

May 17 02:37:18 ubuntu dhcpd[9410]:
May 17 02:37:18 ubuntu dhcpd[9410]: If you think you have received this message due to a bug rather
May 17 02:37:18 ubuntu dhcpd[9410]: than a configuration issue please read the section on submitting
May 17 02:37:18 ubuntu dhcpd[9410]: bugs on either our web page at www.isc.org or in the README file
May 17 02:37:18 ubuntu dhcpd[9410]: before submitting a bug. These pages explain the proper
May 17 02:37:18 ubuntu dhcpd[9410]: process and the information we find helpful for debugging.
May 17 02:37:18 ubuntu dhcpd[9410]:
May 17 02:37:18 ubuntu dhcpd[9410]: exiting.
May 17 02:37:18 ubuntu systemd[1]: isc-dhcp-server.service: Main process exited, code=exited, status=1/FAILURE
May 17 02:37:18 ubuntu systemd[1]: isc-dhcp-server.service: Failed with result 'exit-code'.
lines 1-18/18 (END)

```

Figure 11.2: `isc-dhcp-server` failing by default

If it did fail to start, there's no need to be concerned. Ubuntu, by default, starts most of the services that are installed via packages. Sometimes, a service needs to be configured before it can run. In the case of the `isc-dhcp-server`, it needs a valid configuration in order to start, but we have yet to configure anything. We need to configure the `isc-dhcp-server` service for it to be useful, so let's stop the service for now.

We can start it as soon as we've finished adding our configuration:

```
sudo systemctl stop isc-dhcp-server
```

Configuring an IPv6 network isn't something we'll be getting into, but the DHCP server package we've just installed also comes with an IPv6 equivalent. Let's stop and disable this service, since we won't be using it:

```
sudo systemctl stop isc-dhcp-server6
sudo systemctl disable isc-dhcp-server6
```

Now that you've installed the `isc-dhcp-server` package, you'll have a default configuration file for it at `/etc/dhcp/dhcpd.conf`. This file will contain some default configuration, with some example settings that are commented out. Feel free to take a look at this file to get an idea of some of the settings you can configure. We'll create our own `dhcpd.conf` file from scratch. So when you're done looking at it, move the existing file with a new name so we can refer to it later if we ever need to:

```
sudo mv /etc/dhcp/dhcpd.conf /etc/dhcp/dhcpd.conf.orig
```

Now, we're ready to create our own `dhcpd.conf` file. Open `/etc/dhcp/dhcpd.conf` in your preferred text editor. Since the file no longer exists (we moved it), we should start with an empty file. Here's an example `dhcpd.conf` file that I will explain so that you understand how it works:

```
default-lease-time 43200;
max-lease-time 86400;
option subnet-mask 255.255.255.0;
option broadcast-address 192.168.1.255;
option domain-name "local.lan";
authoritative;
subnet 192.168.1.0 netmask 255.255.255.0 {
    range 192.168.1.100 192.168.1.240;
    option routers 192.168.1.1;
    option domain-name-servers 192.168.1.1;
}
```

As always, change the values I've used to those that match your network. I'll explain each line so that you'll understand how it affects the configuration of your DHCP server.

```
default-lease-time 43200;
```

When a device connects to your network and requests an IP address, the expiration of the lease will be set to the number of seconds in `default-lease-time` if the device doesn't explicitly ask for a longer lease time. Here, I'm setting that to 43200 seconds, which is equivalent to half a day. This basically means that the device will need to renew its IP address every 43200 seconds unless it asks for a longer duration.

```
max-lease-time 86400;
```

While the previous setting dictated the default lease time for devices that don't ask for a specific lease time, `max-lease-time` is the most that the device is allowed to have. In this case, I set this to one day (86400 seconds). Therefore, no device that receives an IP address from this DHCP server is allowed to hold onto its lease for longer than this without first renewing it.

```
option subnet-mask 255.255.255.0;
```

With this setting, we're informing clients that their subnet mask should be set to `255.255.255.0`, which is for a default 24-bit network. If you plan to subnet your network, you'll put in a different value here. `255.255.255.0` is fine if all you need is a 24-bit network.

```
option broadcast-address 192.168.1.255;
```

With this setting, we're telling the client to use `192.168.1.255` as the broadcast address, which is the last address in the subnet and cannot be assigned to a host.

```
option domain-name "local.lan";
```

Here, we're setting the domain names of all hosts that connect to the server to include `local.lan`. The domain name is added to the end of the hostname. For example, if a workstation with a hostname of `muffin` receives an IP address from our DHCP server, it will be referred to as `muffin.local.lan`. Feel free to change this to the domain name of your organization, or you can leave it as is if you don't have one.

```
authoritative;
```

With the `authoritative` setting (the opposite is `not authoritative`), we're declaring our DHCP server as authoritative to our network. Unless you are planning to have multiple DHCP servers, the `authoritative;` option should be included in your config file. We won't use the `not authoritative;` option as it's beyond the scope of this chapter.

Now, we get to the most important part of our configuration file for DHCP. The following block details the specific information that will be provided to clients:

```
subnet 192.168.1.0 netmask 255.255.255.0 {  
    range 192.168.1.100 192.168.1.240;  
    option routers 192.168.1.1;  
    option domain-name-servers 192.168.1.1;  
}
```

This block is probably self-explanatory, but we're basically declaring our pool of addresses for the `192.168.1.0` network. We're declaring a range of IPs from `192.168.1.100` through `192.168.1.240` to be available for clients. Now when our DHCP server provides an address to clients, it will choose one from this pool. For the address pool (range), feel free to expand it or shrink it accordingly. For example, you might need more addresses than the 140 that are allowed in my sample range, so you may change it to something like `192.168.1.50` through `192.168.1.250`. Feel free to experiment.

We're also providing a default gateway (option `routers`) and DNS server (option `domain-name-servers`) of `192.168.1.1`. This is assuming that your router and local DNS server are both listed at that address, so make sure that you change it accordingly. Otherwise, anyone who receives a DHCP lease from your server will not be able to connect to anything.

Now we have our configuration file in place, but the DHCP server will likely still not start until we declare an interface for it to listen for requests on. You can do that by editing the `/etc/default/isc-dhcp-server` file, where you'll see a line toward the bottom similar to the following:

```
INTERFACESv4=""
```

Simply type the name of the interface within the quotes:

```
INTERFACESv4="enp0s3"
```

In case you forgot, the command to list the details of the interfaces on your server is `ip addr show`, or the shortened version, `ip a`.


Now that we have our DHCP server configured, we should be able to start it:

```
sudo systemctl start isc-dhcp-server
```

Next, double-check that there were no errors by running the following command:

```
sudo systemctl status isc-dhcp-server
```

The daemon should report that it's active (running), similar to what's shown in the following screenshot:



```

isc-dhcp-server.service - ISC DHCP IPv4 server
  Loaded: loaded (/lib/systemd/system/isc-dhcp-server.service; enabled; vendor preset: enabled)
  Active: active (running) since Tue 2022-05-17 02:44:49 UTC; 13s ago
    Docs: man:dhcpd(8)
   Main PID: 10302 (dhcpd)
     Tasks: 4 (limit: 9251)
    Memory: 4.5M
       CPU: 11ms
    CGroup: /system.slice/isc-dhcp-server.service
            └─10302 dhcpd -user dhcpd -group dhcpd -f -4 -pf /run/dhcp-server/dhcpd.pid -cf /etc/dhcp/dhcpd.conf

May 17 02:44:49 ubuntu sh[10302]: Wrote 0 leases to leases file.
May 17 02:44:49 ubuntu dhcpd[10302]: PID file: /run/dhcp-server/dhcpd.pid
May 17 02:44:49 ubuntu dhcpd[10302]: Wrote 0 leases to leases file.
May 17 02:44:49 ubuntu dhcpd[10302]: Listening on LPF/enp0s25/f4:4d:30:62:3c:26/172.16.250.0/24
May 17 02:44:49 ubuntu sh[10302]: Listening on LPF/enp0s25/f4:4d:30:62:3c:26/172.16.250.0/24
May 17 02:44:49 ubuntu sh[10302]: Sending on LPF/enp0s25/f4:4d:30:62:3c:26/172.16.250.0/24
May 17 02:44:49 ubuntu sh[10302]: Sending on Socket/fallback/fallback-net
lines 1-18

```

Figure 11.3: Checking the status of the `isc-dhcp-server` process after a successful start

Assuming all went well, your DHCP server should be running. When an IP lease is assigned to a client, it will be recorded in the `/var/lib/dhcp/dhcpd.leases` file. While your DHCP server runs, it will also record information to your server's system log, located at `/var/log/syslog`. To see your DHCP server function in all its glory, you can follow the log as it gets written to with the following:

```
sudo tail -f /var/log/syslog
```

We've discussed the `-f` flag of the `tail` command earlier in the book. This option is indispensable, and it's something you'll likely use quite often as a server administrator. With the `-f` option, you'll watch the log as it gets written to, rather than needing to refresh it manually. You can press `Ctrl + c` to break out of the file.

While your DHCP server runs, you'll see notices appear within the `syslog` file whenever a DHCP request was received and when a lease is offered to a client. A typical DHCP request will appear in the log similar to the following:

```
Oct  5 22:07:36 myserver dhcpd: DHCPDISCOVER from 52:54:00:88:f8:bc via enp0s3
Oct  5 22:07:36 myserver dhcpd: DHCPOFFER on 192.168.1.103 to 51:52:01:87:f7:bc
via enp0s3
```

Active and previous DHCP leases are stored in the `/var/lib/dhcp/dhcpd.leases` file, and a typical lease entry in that file would look similar to the following:

```
lease 192.168.1.138 {
    starts 0 2022/10/05 16:37:30;
    ends 0 2022/10/06 16:42:30;
    cltt 0 2022/20/06 16:37:30;
    binding state active;
    next binding state free;
    rewind binding state free;
    hardware ethernet 32:6e:92:01:1f:7f;
}
```

When a new device is added to your network and receives an IP address from your new DHCP server, you should see the lease information populate that file. This file can be incredibly helpful, because whenever you connect a new device, you won't have to interrogate the device itself to find out what its IP address is. You can just check the `/var/lib/dhcp/dhcpd.leases` file. If the device advertises its hostname, you'll see it within its lease entry. A good example of how this can be useful is connecting a Raspberry Pi to your network. Once you plug it in and turn it on, you'll see its IP address in the `dhcpd.leases` file, and then you can connect to it via SSH with the IP without having to plug a monitor into it to find out which IP it was given. Similarly, you can view the temporary IP address of a new network appliance that you plug in so that you can connect to it and configure it.

If you have any trouble setting up the `isc-dhcp-server` daemon, double-check that you have set all the correct and matching values within your static IP assignment, as well as within your `/etc/dhcp/dhcpd.conf` file. For example, your server must be within the same network as the IPs you're assigning to clients. As long as everything matches, you should be fine and it should start properly.

Next, let's take a look at another important service within a network—DNS.

Adding a DNS server

I'm sure most of you are familiar with the purpose of a **Domain Name System (DNS)** server. Its simplest definition is that it's a service that's responsible for matching an IP address to a domain or hostname. When you're connected to the internet, name-to-IP matching happens constantly as you browse. After all, it's much easier to connect to `https://www.google.com/` using its domain name than it is to remember its IP address.

When you connect to the internet, your workstation or server will connect to an external DNS server in order to figure out the IP addresses for the websites you attempt to visit.

It's also very common to run a local DNS server internally in your organization. The benefit is that you'll be able to resolve your local hostnames as well, something that an external DNS server would know nothing about. For example, if you have an intranet site that you intend to make available to your co-workers, it would be easier to give everyone a local domain that they can access than it would be to make everyone memorize its IP address. With a local DNS server, you would create what is known as a **zone file**, which would contain information regarding the hosts and IP addresses in use within your network so that local devices would be able to resolve them. In the event that your local DNS server is unable to fulfill your request (such as a request for an external website), the server would pass the request along to an external DNS server, which would then carry out the request.

While DNS is an extensive topic that can take a while to master, a basic understanding is really all you need in order to make use of a custom DNS server within your network. In this section, I'll show you how to set up your very own DNS server to allow your devices to resolve local hostnames, which should greatly enhance your network.

Setting up external DNS with bind

To set up your very own DNS server, we'll first need to install the **Berkeley Internet Name Daemon (BIND)** package on our server:

```
sudo apt install bind9
```

Now, we should have the `bind9` service running on our server, though it's not actually configured to do much at this point. The most basic function of `bind` is to act as what's called a **Caching Name Server**, which means that the server doesn't actually match any names itself. Instead, it caches responses from an external server – which can actually increase the performance of your network in some cases. We'll configure `bind` with actual hosts later, but setting up a caching name server is a good way to get started.

To do so, open the `/etc/bind/named.conf.options` file in your favorite text editor.

Within the file, you should see a block of text that looks similar to the following:

```
// forwarders {  
//     0.0.0.0;  
// };
```

Uncomment these lines. The *forward slashes* are comment marks as far as this configuration file is concerned, so remove them. Then, we can add a few external DNS server IP addresses. For these, you can use the IP addresses for your **Internet Service Provider (ISP)**'s DNS servers, or you could simply use Google's DNS servers (8.8.8.8 and 8.8.4.4) instead:

```
forwarders {  
    8.8.8.8;  
    8.8.4.4;  
};
```


After you save the file, restart the `bind9` service:

```
sudo systemctl restart bind9
```

To be sure that everything is running smoothly, check the status of the service:

```
systemctl status bind9
```

It should report that it's active (running). As long as you've entered everything correctly, you should now have a working DNS server. Of course, we haven't added any DNS names for it to resolve, but we'll get to that. Now, all you should need to do is configure other devices on your network to use your new DNS server. The easiest way to do this is to reconfigure the `isc-dhcp-server` service we set up in the previous section. Remember the section that designates a pool of addresses from the server to the clients? It also contained a section to declare the DNS server your clients will use as well. Here's that section again, with the relevant line in bold:

```
subnet 192.168.1.0 netmask 255.255.255.0 {  
    range 192.168.1.100 192.168.1.240;  
    option routers 192.168.1.1;  
    option domain-name-servers 192.168.1.1;  
}
```

To configure the devices on your network to use your new DNS server, all you should need to do is change the configuration option `domain-name-servers 192.168.1.1`; to point to the IP address of the new DNS server that we're in the process of setting up. When clients request a DHCP lease (or attempt to renew an existing lease), they will be configured with the new DNS server automatically.

With the caching name server we just set up, hosts that utilize it will check it first for any hostnames they attempt to look up. If they look up a website or host that is not within your local network, their requests will be forwarded to the forwarding addresses you configured for `bind`.

In my example, I used Google's DNS servers, so if you used my configuration, your hosts will first check your local server and then check Google's servers when resolving external names. Depending on your network hardware and configuration, you might even see a slight performance boost. This is because the DNS server you just set up is caching any lookups done against it. For example, if a client looks up `https://www.packtpub.com` in a web browser, your DNS server will forward the request along since that site doesn't exist locally and it will also remember the result. The next time a client within your network looks up that site, the response will be much quicker because your DNS server cached it.

To see this yourself, execute the following command twice on a node (device) that is utilizing your new DNS server:

```
dig www.packtpub.com
```

If `dig` isn't available in your installation, you can install it as part of the `dnsutils` package:

```
sudo apt install dnsutils
```

In the response, look for a line toward the end that gives you your query time. It will look similar to the following:

```
;; Query time: 98 msec
```

When you run it again, the query time should be much lower:

```
;; Query time: 1 msec
```

This is your caching name server in action! Even though we haven't even set up any zone files to resolve your internal servers, your DNS server is already adding value to your network. You just laid the groundwork we'll use for the rest of our configuration.

Setting up internal DNS and adding hosts

Now, let's add some hosts to our DNS server so we can start fully utilizing it. The configuration file for `bind` is located at `/etc/bind/named.conf`. In addition to some commented lines, it will have the following three lines of configuration within it:

```
include "/etc/bind/named.conf.options";
include "/etc/bind/named.conf.local";
include "/etc/bind/named.conf.default-zones";
```

As you can see, the default `bind` configuration is split among several configuration files. Here, it includes three others: `named.conf.options`, `named.conf.local`, and `named.conf.default-zones` (the first of which we already took care of editing). In order to resolve local names, we need to create a **zone file**, which is essentially a text file that includes some configuration, a list of hosts, and their IP addresses. In order to do this, we need to tell `bind` where to find the zone file we're about to create. Within `/etc/bind/named.conf.local`, we need to add a block of code like the following to the end of the file:

```
zone "local.lan" IN {
    type master;
    file "/etc/bind/net.local.lan";
};
```

Notice that the zone is named `local.lan`, which is the same name I gave our domain in our DHCP server configuration. It's best to keep everything consistent when we can. If you use a different domain name than the one I used in my example, make sure that it matches here as well. Within the block, we're creating a master zone file and informing `bind` that it can find a file named `net.local.lan`, stored in the `/etc/bind` directory. This should be the only change we'll need to make to the `named.conf.local` file; we'll only create a single zone file (for the purpose of this section). Once you save this file, you'll need to create the `/etc/bind/net.local.lan` file. So, go ahead and open that file in a text editor. Since we haven't created it yet, it should be blank. Here's an example of this zone file, completely filled out with some sample configuration:

```
$TTL 1D
@ IN SOA local.lan. hostmaster.local.lan. (
```

```

202208161; serial

8H ; refresh
4H ; retry
4W ; expire
1D ) ; minimum
IN A 192.168.1.1
;
@ IN NS hermes.local.lan.
fileserv      IN  A   192.168.1.3
hermes        IN  A   192.168.1.1
mailserv      IN  A   192.168.1.5
mail          IN  CNAME mailserv.
web01         IN  A   192.168.1.7

```

Feel free to edit this file to match your configuration. You can edit the list of hosts at the end of the file to match your hosts within your network, as the ones I included are merely examples. You should also ensure that the file matches the IP scheme for your network. Next, I'll go over each line in order to give you a deeper understanding of what each line of this configuration file is responsible for:

```
$TTL 1D
```

The **Time to Live (TTL)** determines how long a record may be cached within a DNS server. If you recall from earlier, where we practiced with the `dig` command, you saw that the second time you queried a domain with `dig`, the query time was less than the first time you ran the command. This is because your DNS server cached the result, but it won't hold onto it forever. At some point, the lookup will expire. The next time you look up that same domain after the cached result has expired, your server will go out and fetch the result from the DNS server again. In my examples, I used Google's DNS servers. That means at some point, your server will query those servers again once the record times out.

With the **Start of Authority (SOA)** line, we're establishing that our DNS server is authoritative over the `local.lan` domain:

```
@ IN SOA local.lan. hostmaster.local.lan. (
```

We also set `hostmaster@local.lan` as the email address of the responsible party for this server, but we enter it here in a different format for `bind` (`hostmaster.local.lan`). This is obviously a fake address, but for the purposes of an internal DNS server, its validity is of no concern.

Of all the lines of configuration within a zone file, `serial` is by far the one that will frustrate us the most:

```
202208161; serial
```

This is because it's not enough to simply update the zone file any time we make a change to it (change an IP address, add or remove a host, and so on); we also need to remember to increase the serial number by at least one. If we don't, `bind` won't be aware that we've made any changes, as it will look at the serial number before the rest of the file.

The problem with this is that you and I are both human, and we're prone to forgetting things. I've forgotten to update `serial` many times and have become frustrated when the DNS server refused to resolve new hosts that were recently added. Once I remembered that I didn't increment the serial number, the issue was resolved after I did. Therefore, it's very important for you to remember that any time you make a change to any zone file, you'll need to also increment the serial number.

The format doesn't really matter; I used `202208161`, which is simply the year, two-digit month, two-digit day, and an extra number to cover us if we make more than one change in a day (which can sometimes happen). As long as you increment the serial number by one every time you modify your zone file, you'll be in good shape—regardless of what format you use. However, the sample format I gave here is actually quite common in the field.

These values control how often secondary DNS servers will be instructed to check in for updates:

```
8H ; refresh
4H ; retry
4W ; expire
1D ) ; minimum
```

With the example `refresh` value, we're instructing any secondary DNS servers to check in every eight hours to see whether or not the zone records were updated. The `retry` field dictates how long the secondary will wait to check in, in case there was an error doing so the last time. The last two options in this section, `expire` and `minimum`, set the minimum and maximum age of the zone file, respectively. As I mentioned though, a full discussion of DNS with `bind` could constitute an entire book on its own. For now, I would just use these values until you have a reason to need to experiment. Here, we identify the name server itself:

```
IN A 192.168.1.1
@ IN NS hermes.local.lan.
```

In my case, the server is called `hermes` and it's located at `192.168.1.1`.

Next, in our file, we'll have several host entries to allow our resources to be resolved on our network by name:

```
fileserv      IN A    192.168.1.3
hermes        IN A    192.168.1.1
mailserv      IN A    192.168.1.5
mail          IN CNAME mailserv.
web01         IN A    192.168.1.7
```

In that example, I have three hosts: `fileserv`, `mailserv`, and `web01`. In the example, these are all address records, which means that any time our server is asked to resolve one of these names, it will respond with the corresponding IP address. If our DNS server is set as a machine's primary DNS server, it will respond with `192.168.1.3` when asked for `fileserv` and `192.168.1.7` when asked for `web01`.

The entry for `mail` is special as it is not an address record, but instead a **Canonical Name (CNAME)** record. In this case, it just points back to `mailserv`. Essentially, that's what a CNAME record does: it creates a pointer to another resource. In this case, if someone tries to access a server named `mail`, we redirect them to the actual server `mailserv`. Notice that on the CNAME record, we're not inputting an IP address, but instead the hostname of the resource it's linked to.

In addition, you should also notice that I added the DNS server itself (`hermes`) to the file as well. You can see it on the second line above. I've found that if you don't do this, the DNS server may complain and refuse to load the file.

Now that we have a zone file in place, we should be able to start using it. First, we'll need to restart the `bind9` service:

```
sudo systemctl restart bind9
```

After the command finishes, check to see if there are any errors:

```
systemctl status bind9
```

You should see that the service state is `active (running)`, and in addition, you should see a line telling you that the serial number for your zone file was loaded. If you see that the service is not running and/or your zone file was not loaded, you should see specific information in the output while checking the status that should point you in the right direction. If not, you can also check the system log for clues regarding `bind` as well:

```
cat /var/log/syslog | grep bind9
```

The most common mistakes I've seen typically result from not being consistent within the file. For example, if you're using a different IP scheme (such as `10.10.10.0/24`), you'll want to make sure you didn't forget to replace any of my example IP addresses with the proper scheme. Assuming that everything went smoothly, you should be able to point devices on your network to use this new DNS server. Make sure you test not only pinging devices local to your network but outside resources as well, such as websites. If the DNS server is working properly, it should resolve your local names, and then forward your requests to your external DNS servers (the two we set as forwarders) if it doesn't find what you're looking for locally. In addition, you'll also want to make sure that port 53 (UDP) is open in your network's firewall, which is the port that DNS uses. It's extremely rare that this would be an issue, but I have seen it happen.

To further test our DNS server, we can use the `dig` command, as we did before while we were experimenting with caching. Try `dig` against a local server on your LAN, as well as a DNS address that's not located on your LAN (change the first domain to an actual domain on your LAN):

```
dig webserv.local.lan
dig www.packtpub.com
```

You should see a response similar to the following:

```
;; Query time: 1 msec
;; SERVER: 127.0.0.53#53(127.0.0.53)
```

```
;; WHEN: Sat Feb 10 10:00:59 EST 2022
;; MSG SIZE rcvd: 83
```

What you're looking for here is for both local resources and external websites to be resolvable now. You'll probably notice that the DNS server used in the output will most likely show up as a localhost address, as it did in my output, and not the address of the DNS server we just set up. Actually, you can ignore this. Most distributions of Linux nowadays use local resolvers, which essentially cache DNS lookup results on your local computer. Your computer is still using the DNS server we set up, but there's just an additional layer in between your computer and the DNS server. You can verify this with the following command:

```
resolvectl
```

The output will show you the IP address of the actual server that's responding to your DNS lookups.

Next, let's take a look at the process of setting up an internet gateway, which is an option to consider if you don't already have a router or firewall on your network that acts as a device between your internet connection and your internal network.

Setting up an internet gateway

As long as we're setting up network services, we may as well go all the way and set up a router to act as a **gateway** for our network. A gateway within a network is the device you go through to route from one network to another. In this context, the **internet gateway** will be the device that sits between your local network and the device that provides your internet connection (such as a cable modem). The gateway in a typical network is usually a commercial router or firewall, which often also provides DNS, DHCP, and routing services as well.

If you already have such a device on your network providing these services, then there's nothing for you to do. You can skip this section. But if you'd like to set up your own router, then feel free to proceed.

If you'd like to proceed and set up a router, then the first order of business is to decide which device on your network will serve that purpose. Often, administrators will build DNS, DHCP, and routing services all into the same server, so you can even use the same device you've used earlier to work through the DNS and DHCP examples for our purposes in this section. In order for a device to function as a gateway, it should have at least two network interfaces, one to your ISP device (such as a cable modem) and another interface connected to a network switch that your other servers will connect to. The interface connected to your ISP device should use DHCP, so it will obtain an IP address directly from your ISP. This interface may need a static IP with details provided from the ISP, if relevant.

Depending on what kind of internet connection you have, Linux itself can likely replace whatever device your internet modem connects to. A good example of this is a cable modem that your office or home router may utilize. In this case, the modem provides your internet connection, and then your router allows other devices on your network to access it. In some cases, your modem and router may even be the same device. Therefore, depending on the hardware you have, this method of setting up your networking may or may not be efficient. But if you do have the hardware available, you'll be able to manage the entire networking stack with Ubuntu Server quite easily.

Why might you want to create your own internet gateway? One potential reason is that it's often the case that security patches aren't provided for commercial routers and firewall devices. New vulnerabilities are discovered all the time, and if your router or firewall is no longer supported by the manufacturer, it may allow outside threats into your network. By setting up an internet gateway with Ubuntu, you'll benefit from the regular updates that Canonical provides. As long as you're using a version of Ubuntu that's still supported (such as an LTS release, as is the case with Ubuntu 22.04), you'll benefit from a more secure platform. If nothing else, a physical server that we can install Ubuntu on will usually have a more powerful CPU than a commercial device would have, which would mean that CPU bottlenecks slowing down network performance would be less likely to happen.

Thankfully, setting up an internet gateway is easy. In fact, we'll only need to execute a single command to set up routing between interfaces, which is technically all that's required in order to set up an internet gateway. But before we get into that, it's also important to keep in mind that if you do set up an internet gateway, you'll need to pay special attention to security. The device that sits between your network and your modem will be a constant attack target, just like any other gateway device would be. When it comes to commercial routers, they're also attacked constantly.

However, in most cases, they'll have some sort of default security or firewall built in. In all honesty, the security features built into common routing equipment are extremely poor and most of them are easy to hack when someone wants to badly enough. The point is that these devices have some sort of security to begin with (regardless of how good or bad), whereas a custom internet gateway of your own won't have any security at all until you add it.

When you set up an internet gateway, you'll want to pay special attention to setting up the firewall, restricting access to SSH, using very strong passwords, keeping up to date on security patches, and installing an authentication monitor such as `fail2ban`. We'll get into those topics in *Chapter 21, Securing Your Server*. The reason I bring this up now, though, is that if you do set up an internet gateway, you'll probably want to take a detour and read that chapter right away, just to make sure that you secure it properly.

Anyway, let's move on. A proper internet gateway, as I've mentioned, will have two Ethernet ports. On the first, you'll plug in your cable modem or internet device, and you'll connect a switch to the second. By default though, routing between these interfaces will be disabled, so traffic won't be able to move from one Ethernet port to the other. To rectify this, use the following command:

```
echo 1 | sudo tee /proc/sys/net/ipv4/ip_forward
```

That's actually it. With that single command, you've just made your server into a router. However, that change will not survive a reboot. To make it permanent, open the `/etc/sysctl.conf` file in your editor:

```
sudo nano /etc/sysctl.conf
```

Look for the following line:

```
#net.ipv4.ip_forward=1
```

Uncomment the line by removing the hash symbol in front of it, and save the file. With that change made, your server will allow routing between interfaces even after a reboot. Of all the topics we've covered in this chapter, that one was probably the simplest. However, I must remind you again to definitely secure your server if it's your frontend device to the internet, as computer security students always enjoy practicing on a real-life Linux server. With good security practices, you'll help ensure that they'll leave you alone, or at least have a harder time breaking in.

From here, all you should need to do is attach a network switch to your other network interface, and then you can attach your other wired Ethernet devices and wireless access point to the switch. Now, Ubuntu Server is managing your entire network!

Summary

In this chapter, we explored additional networking topics. We started off with some notes on planning an IP address scheme for your network so that you could create groups for the different types of nodes, such as servers and network equipment, and plan a pool of addresses for DHCP. We also worked through the process of setting up a DHCP and DNS server, which gives us additional flexibility when configuring the services we run on our network, such as when defining a custom IP scheme for DHCP, as well as giving us the ability to resolve the hostnames of devices on our network by name. We closed off this chapter with a discussion on setting up an internet gateway to serve as our internet-facing router.

In the next chapter, we'll take a look at sharing and transferring files over the network. This will include covering NFS and Samba shares, as well as using `scp`, `rsync`, and `sshfs`. Stay tuned!

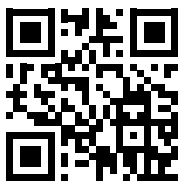
Further reading

- 8 Steps to Understanding IP Subnetting: <https://learnlinux.link/8-ip-sub>
- IP Subnet Calculator: <https://learnlinux.link/s-calc>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



12

Sharing and Transferring Files

In the previous chapter, we looked at what's involved in the process of setting up a few network services, such as DHCP and DNS. Those are two important components of a network, but there are quite a few different types of applications and resources you can make available on your network to further enhance it. A file server is one such example, which can give your users a central place to store critical files and can even enhance collaboration.

Perhaps you've used a file server before, or even set one up on a different platform. With Ubuntu Server, there are multiple methods to not only store files, but also to transfer files from one node to another over a network link. In this chapter, we'll look into setting up a central file server using both Samba and NFS, as well as how to transfer files between nodes with utilities such as `scp` and `rsync`. We'll also go over some situations in which one solution may work better for you than another. As we go through these concepts, we will cover the following topics:

- File server considerations
- Sharing files with Windows users via Samba
- Setting up NFS shares
- Transferring files with `rsync`
- Transferring files with `scp`

Before we can get started with configuring our server to enable it to share files with other users, we should first understand what our available options are to enable us to choose the best technology for our use case.

File server considerations

When it comes to setting up a file server, the process is a matter of setting up some sort of daemon to accept connections and share specific directories, and ensure the appropriate users are able to access those directories. You'll also implement permissions to determine who can access specific directories, and what type of access they will have (read/write, read-only, and so on). When deciding *how* to share the files, it's generally a choice between two common technologies that can facilitate the actual sharing, **Samba** and **NFS**.

All in all, there's nothing stopping you from hosting both Samba and NFS shares on a single server. The two technologies can actually co-exist on the same device. However, each of the two popular solutions is more appropriate for particular use cases. Before we get started with setting up our file server, we should first understand the differences between Samba and NFS, so we can make an informed decision as to which one is more appropriate for our environment. As a general rule of thumb, Samba is great for mixed environments (where you have Windows as well as Linux clients), and NFS is more appropriate for use in Linux or Unix environments, but there's a bit more to it than that.

Samba is a great solution for many environments, because it allows you to share files with Windows, Linux, and macOS machines. Basically, pretty much everyone will be able to access your shares, provided you give them permission to do so. The reason this works is because Samba is a re-implementation of the **Server Message Block (SMB)** protocol, which is primarily used by Windows systems. However, you don't need to use the Windows platform in order to be able to access Samba shares, since many platforms offer support for this protocol.

You may be wondering why I am going to cover two different solutions in this chapter. After all, if Samba shares can be accessed by pretty much everything and everyone, why bother with anything else? Even with Samba's many strengths, there are weaknesses as well. First of all, permissions are handled very differently, so you'll need to configure your shares in specific ways in order to prevent access to users that shouldn't be able to see confidential data. With NFS, full support for standard UNIX permissions is provided, so you'll only need to configure your permissions once. If permissions and confidentiality are important to you, you may want to look closer at NFS.

It's also not accurate to say that Windows systems cannot access NFS shares, because some versions actually can. By default, no version of Windows supports NFS outright, but some editions offer a plugin you can install that enables this support. If you install **Services for NFS**, your installation of Windows will be able to access NFS shares from that point forward.

Regarding an all-Linux environment or a situation where you only have Linux machines that need to access your shares, NFS is a great choice because it integrates much more tightly with the rest of the distribution. Permissions can be more easily enforced and, depending on your hardware, performance may be higher. The specifics of your environment will make your decision for you. Perhaps you'll choose Samba for your mixed environment, or NFS for your all-Linux environment. Maybe you'll even set up both NFS and Samba, having shares available for each platform. My recommendation is to learn and practice both, since you'll use both solutions at one point or another during your career anyway.

Before you continue to the sections on setting up Samba and NFS, I recommend you first decide where in your filesystem you'd like to act as a parent directory for your file shares. This isn't actually required, but I think it makes for better organization. There is no one right place to store your shares, but personally I like to create a `/share` directory at the root filesystem and create sub-directories for my network shares within it. For example, I can create `/share/documents`, `/share/public`, and so on for Samba shares. With regard to NFS, I usually create shared directories within `/exports`. You can choose how to set up your directory structure. As you read the remainder of this chapter, make sure to change my example paths to match yours if you use a different style.

Sharing files with Windows users via Samba

In this section, I'll walk you through setting up your very own **Samba** file server. I'll also go over a sample configuration to get you started so that you can add your own shares. First, we'll need to make sure that the `samba` package is installed on our server:

```
sudo apt install samba
```

When you install the `samba` package, you'll have a new daemon installed on your server, `smbd`. The `smbd` daemon will be automatically started and enabled for you. You'll also be provided with a default configuration file for Samba, located at `/etc/samba/smb.conf`. For now, I recommend stopping `samba` since we have yet to configure it:

```
sudo systemctl stop smbd
```

Since we're going to configure Samba from scratch, we should start with an empty configuration file. Let's back up the original file, rather than overwrite it. The default file includes some useful notes and samples, so we should keep it around for future reference:

```
sudo mv /etc/samba/smb.conf /etc/samba/smb.conf.orig
```

Now, we can begin a fresh configuration. Although it's not required, I like to split my Samba configuration up between two files, `/etc/samba/smb.conf` and `/etc/samba/smbshared.conf`. You don't have to do this, but I think it makes the configuration cleaner and easier to read. First, here is a sample `/etc/samba/smb.conf` file:

```
[global]
server string = File Server
workgroup = WORKGROUP
security = user
map to guest = Bad User
name resolve order = host bcast wins
include = /etc/samba/smbshared.conf
```

As you can see, this is a really short file. Basically, we're including only the lines we absolutely need to in order to set up a file server with Samba. Next, I'll explain each line and what it does.

```
[global]
```

With the `[global]` stanza, we're declaring the global section of our configuration, which will consist of settings that will impact Samba as a whole. There will also be additional stanzas for individual shares, which we'll get to later.

```
server string = File Server
```

The `server` string is somewhat of a description field for the File Server. If you've browsed networks from Windows computers before, you may have seen this field. Whatever you type here will display underneath the server's name in Windows Explorer. This isn't required, but it's nice to have.

```
workgroup = WORKGROUP
```

Here, we're setting the workgroup, which is the exact same thing as a workgroup on Windows PCs. In short, the workgroup is a namespace that describes a group of machines. When browsing network shares on Windows systems, you'll see a list of workgroups, and then one or more computers within that workgroup. In short, this is a way to logically group your nodes. You can set this to whatever you like. If you already have a workgroup in your organization, you should set it here to match the workgroup names of your other machines. The default workgroup name is simply `WORKGROUP` on Windows PCs, if you haven't customized the workgroup name at all.

```
security = user
```

This setting sets up Samba to utilize usernames and passwords for authentication to the server. Here, we're setting the `security` mode to `user`, which means we're using local users to authenticate, rather than other options such as `ads` (Active Directory) or `domain` (domain controller), which are both outside the scope of this book.

```
map to guest = Bad User
```

This option configures Samba to treat unauthenticated users as guest users. Basically, unauthenticated users will still be able to access shares, but they will have guest permissions instead of full permissions. If that's not something you want, then you can omit this line from your file. Note that if you do omit this, you'll need to make sure that both your server and client PCs have the same user account names on either side. Ideally, we want to use directory-based authentication, but that's beyond the scope of this book.

```
name resolve order = host bcast wins
```

The `name resolve order` setting configures how Samba resolves hostnames. In this case, we're using the broadcast name first, followed by `wins`. Since `wins` has been pretty much abandoned (and replaced by DNS), we include it here solely for compatibility with legacy networks.

```
include = /etc/samba/smbshared.conf
```

Remember how I mentioned that I usually split my Samba configurations into two different files? On this line, I'm calling that second `/etc/samba/smbshared.conf` file. The contents of the `smbshared.conf` file will be inserted right here, as if we only had one file. We haven't created the `smbshared.conf` file yet. Let's take care of that next. Here's a sample `smbshared.conf` file:

```
[Documents]
path = /share/documents
force user = myuser
```

```
force group = users
public = yes
writable = no

[Public]
path = /share/public
force user = myuser
force group = users
create mask = 0664
force create mode = 0664
directory mask = 0777
force directory mode = 0777
public = yes
writable = yes
```

As you can see, I'm separating share declarations in their own file. We can see several interesting things within `smbshred.conf`. First, we have two stanzas, `[Documents]` and `[Public]`. Each stanza is a share name, which will allow Windows users to access the share under `//servername/share-name`. In this case, this file will give us two shares: `//servername/Documents` and `//servername/Public`. The `Public` share is writable for everyone, though the `Documents` share is restricted to read-only. The `Documents` share has the following options:

```
path = /share/documents
```

This is the path to the share, which must exist on the server's filesystem. In this case, when a user reads files from `//servername/Documents` on a Windows system, they will be reading data from `/share/documents` on the Ubuntu server that's housing the share.

```
force user = myuser
force group = users
```

These two lines are basically bypassing user ownership. When a user reads this share, they are treated as `myuser` instead of their actual user account. Normally, you would want to set up LDAP or Active Directory to manage your user accounts and handle their mapping to Ubuntu Server, but a full discussion of directory-based user access isn't covered in this book, so I provided the `force` options as an easy starting point. The user account you set here must exist on the server.

```
public = yes
writable = no
```

With these two lines, we're configuring what users are able to do once they connect to this share. In this case, `public = yes` means that the share is publicly available, though `writable = no` prevents anyone from making changes to the contents of this share. This is useful if you want to share files with others, but you want to restrict access and stop anyone from being able to modify the content.

The Public share has some additional settings that weren't found in the Documents share:

```
create mask = 0664
force create mode = 0664
directory mask = 0777
force directory mode = 0777
```

With these options, I'm setting up how the permissions of files and directories will be handled when new content is added to the share. Directories will be given 777 permissions and files will be given permissions of 664. Yes, these permissions are very open; note that the share is named `Public`, which implies full access anyway, and its intended purpose is to house data that isn't confidential or restricted:

```
public = yes
writable = yes
```

Just as I did with the previous share, I'm setting up the share to be publicly available, but this time I'm also configuring it to allow users to make changes.

To take advantage of this configuration, we need to start the Samba daemon. Before we do though, we want to double-check that the directories we entered into our `smbshared.conf` file exist, so if you're using my example, you'll need to create `/share/documents` and `/share/public`:

```
sudo mkdir /share
sudo mkdir /share/documents
sudo mkdir /share/public
```

Also, the user account that was referenced in the `force user` and the group referenced in the `force group` must both exist and have ownership over the shared directories:

```
sudo chown -R jay:users /share
```

At this point, it's a good idea to use the `testparm` command, which will test the syntax of our Samba configuration files for us. It won't necessarily catch every error we could have made, but it is a good command to run to quickly check the sanity. This command will first check the syntax, and then it will print the entire configuration to the terminal for you to have a chance to review it. If you see no errors here, then you can proceed to start the service:

```
sudo systemctl start smbd
```

Then, check the status to ensure that the daemon is running:

```
sudo systemctl status smbd
```

This will produce output similar to the following:

```
jay@LearnLinuxTV: ~  
Active: active (running) since Mon 2022-05-23 10:02:09 EDT; 5s ago  
Docs: man:smbd(8)  
      man:samba(7)  
      man:smb.conf(5)  
Process: 5657 ExecStartPre=/usr/share/samba/update-apparmor-samba-profile (code=exited, status=0/SUCCESS)  
Main PID: 5661 (smbd)  
Status: "smbd: ready to serve connections..."  
Tasks: 5 (limit: 9321)  
Memory: 9.9M  
CPU: 123ms  
CGroup: /system.slice/smbd.service  
├─5661 /usr/sbin/smbd --foreground --no-process-group  
├─5663 /usr/sbin/smbd --foreground --no-process-group  
├─5664 /usr/sbin/smbd --foreground --no-process-group  
├─5665 /usr/lib/x86_64-linux-gnu/samba/samba-bgqd --ready-signal-fd=46 --parent-watch-fd=12 --d  
└─5667 /usr/sbin/smbd --foreground --no-process-group  
May 23 10:02:09 LearnLinuxTV systemd[1]: Starting Samba SMB Daemon...  
lines 3-20/25 63%
```

Figure 12.1: Checking the status of the `smbd` daemon

That really should be all there is to it; you should now have Documents and Public shares on your file server that Windows users should be able to access. In fact, your Linux machines should be able to access these shares as well. On Windows, **Windows Explorer** has the ability to browse file shares on your network. If in doubt, try pressing the *Windows* key and the *r* key at the same time to open the **Run** dialog box, and then type the **Universal Naming Convention (UNC)** path to the share (`\\servername\Documents` or `\\servername\Public`). You should be able to see any files stored in either of those directories. In the case of the Public share, you should be able to create new files there as well.

On Linux systems, if you have a desktop environment installed, most of them feature a file manager that supports browsing network shares. Since there are a handful of different desktop environments available, the method varies from one distribution or configuration to another. Typically, most Linux file managers will have a network link within the file manager, which will allow you to easily browse your local shares. In the case of Ubuntu itself, you may need to install `smbclient` and `cifs-utils` via `apt`, so if you get an error you can try installing those packages. In the following screenshot, an Ubuntu 22.04 desktop client is browsing shares on a local Samba server:

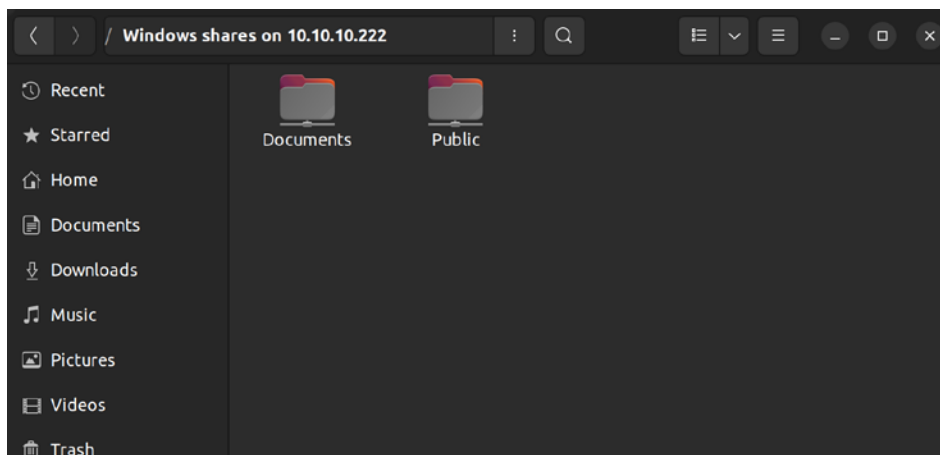


Figure 12.2: Browsing a Samba share from a Linux client

If your file manager doesn't show you the available shares on your server, you can also access a Samba share by adding an entry for it in the `/etc/fstab` file, such as the following:

```
//myserver/share/documents /mnt/documents cifs username=myuser,noauto 0 0
```

Then, assuming the local directory exists (`/mnt/documents` in the example `fstab` line), you should be able to mount the share with the following command:

```
sudo mount /mnt/documents
```

In the `fstab` entry, I included the `noauto` option so that your system won't mount the Samba share at boot time (you'll need to do so manually with the `mount` command). If you do want the Samba share automatically mounted at boot time, change `noauto` to `auto`. However, you may receive errors during boot if for some reason the server hosting your Samba shares isn't accessible, which is why I prefer the `noauto` option.

If you'd prefer to mount the Samba share without adding an `fstab` entry, the following example command should do the trick; just change the share name and mount point to match your local configuration:

```
sudo mount -t cifs //myserver/Documents -o username=myuser /mnt/documents
```

From here, feel free to experiment. You can add additional shares as appropriate, and customize your Samba implementation as you see fit. In the next section, we'll explore NFS.

Setting up NFS shares

An alternative to Samba is NFS. It's a great method of sharing files from a Linux or Unix server to another Linux or Unix server. As I mentioned earlier in the chapter, Windows systems can access NFS shares as well, but that requires an add-on to be enabled. Therefore, NFS is preferred in a Linux or Unix environment, since it fully supports Linux and Unix-style permissions. As you can see from our dive into Samba earlier, we essentially forced all shares to be treated as being accessed by a particular user, which was messy, but was the easiest example of setting up a Samba server without also walking you through setting up a complicated Windows Active Directory deployment. Samba can certainly support per-user access restrictions and benefits greatly from a centralized directory server, though that would basically be a book of its own! NFS integrates better in a non-mixed environment.

Earlier, we set up a parent directory in our filesystem to house our Samba shares, and we should do the same thing with NFS. While it wasn't mandatory to have a special parent directory with Samba (I had you do that in order to be neat, but you weren't required to), NFS really does want its own directory to house all of its shares. It's not strictly required with NFS either, but there's an added benefit in doing so, which I'll go over before the end of this section. In my case, I'll use `/exports` as an example, so you should make sure that that directory, or whatever you've chosen for NFS, exists:

```
sudo mkdir /exports
```

Next, let's install the required NFS packages on our server. The following command will install NFS and its dependencies:

```
sudo apt install nfs-kernel-server
```

Once you install the `nfs-kernel-server` package, the `nfs-kernel-server` daemon will start up automatically. It will also create a default `/etc/exports` file (which is the main file that NFS reads its share information from), but it doesn't contain any useful settings, just some commented lines. Let's back up the `/etc/exports` file, since we'll be creating our own:

```
sudo mv /etc/exports /etc/exports.orig
```

To set up NFS, let's first create some directories that we will share with other users. Each share in NFS is known as an **export**. I'll use the following directories as examples, but you can export any directory you like:

```
/exports/backup
/exports/documents
/exports/public
```

In the `/etc/exports` file (which we're creating fresh), I'll insert the following four lines:

```
/exports *(ro,fsid=0,no_subtree_check)
/exports/backup 192.168.1.0/255.255.255.0(rw,no_subtree_check)
/exports/documents 192.168.1.0/255.255.255.0(ro,no_subtree_check)
/exports/public 192.168.1.0/255.255.255.0(rw,no_subtree_check)
```

The first line is our **export root**, which I'll go over a bit later. The next three lines are individual shares or exports. The `/backup`, `/documents`, and `/public` directories are being shared from the `/exports` parent directory. In this case, after the directory is called out in a line, we're also setting which network is able to access them (`192.168.1.0/255.255.255.0` in our case). This means that if you're connecting from a different network, your access will be denied. Each connecting machine must be a member of the `192.168.1.0/24` network in order to proceed (so make sure you change this to match your IP scheme). Finally, we include some options for each export, for example, `rw,no_subtree_check`.

As far as what these options do, the first (`rw`) is rather self-explanatory. Here, we can set whether or not other nodes will be able to make changes to data within the export. In the examples I gave, the `/documents` export is read-only (`ro`), while the others allow read and write.

The next option in each example is `no_subtree_check`. This option is known to increase reliability and is mainly implied by default. However, not including it may make NFS complain when it restarts, but nothing that will actually stop it from working. Particularly, this option disables what is known as **subtree checking**, which has had some stability issues in the past. Normally, when a directory is exported, NFS might scan parent directories as well, which is sometimes problematic, and can cause issues when it comes to open file handles.

There are several other options that can be included in an export, and you can read more about them by checking the man page for export:

```
man exports
```

One option you'll see quite often in the wild is `no_root_squash`. Normally, the `root` user on one system is mapped to `nobody` on the other for security reasons. In most cases, one system having `root` access to another is a bad idea. The `no_root_squash` option disables this, and it allows the `root` user on one end to be treated as the `root` user on the other. I can't think of a reason, personally, where this would be useful (or even recommended), but I have seen this option quite often in the wild, so I figured I would bring it up. Again, check the man pages for `export` for more information on additional options you can pass to your exports.

Next, we have one more file to edit before we can actually seal the deal on our NFS setup. The `/etc/idmapd.conf` file is necessary for mapping permissions on one node to another. In *Chapter 2, Managing Users and Permissions*, we talked about the fact that each user has an ID (UID) assigned to them. The problem, though, is that from one system to another, a user will not typically have the same UID. For example, user `jdoe` may be UID `1001` on server A, but `1007` on server B. When it comes to NFS, this greatly confuses the situation, because UIDs are used in order to reference permissions. Mapping IDs with `idmapd` allows this to stay consistent and handles translating each user properly, though it must be configured correctly and consistently on each node. Basically, as long as you use the same domain name on each server and client and configure the `/etc/idmapd.conf` file properly on each, you should be fine.

To configure this, open `/etc/idmapd.conf` in your text editor. Look for an option that is similar to the following:

```
# Domain = localdomain
```

First, remove the `#` symbol from that line to uncomment it. Then, change the domain to match the one used within the rest of your network. You can leave this as it is as long as it's the same on each node, but if you recall from *Chapter 11, Setting Up Network Services*, we used a sample domain of `local.lan` in our DHCP configuration, so it's best to make sure you use the same domain name everywhere—even the domain provided by DHCP. Basically, just be as consistent as you can and you'll have a much easier time overall. You'll also want to edit the `/etc/idmapd.conf` file on each node that will access your file server, to ensure they are configured the same as well.

With our `/etc/exports` and `/etc/idmapd.conf` files in place, and assuming you've already created the exported directories on your filesystem, we should be all set to restart NFS to activate our configuration:

```
sudo systemctl restart nfs-kernel-server
```

After restarting NFS, you should check the daemon's output via `systemctl` to ensure that there are no errors:

```
systemctl status nfs-kernel-server
```

As long as there are no errors, our NFS server should be working. Now, we just need to learn how to mount these shares on another system. Unlike Samba, using a Linux file manager and browsing the network will not show NFS exports by default; we'll need to mount them manually. Client machines, assuming they are Debian-based (Ubuntu fits this description), will need the `nfs-common` package installed in order to access these exports.

It may already be installed, but if it's not, we can install it with apt like any other package:

```
sudo apt install nfs-common
```

With the client installed, we can now use the mount command to mount NFS exports on a client. For example, with regards to our documents export, we can use the following variation of the mount command to do the trick:

```
sudo mount myserver:/documents /mnt/documents
```

Replace myserver with either your server's hostname or its IP address, documents with the name of the actual share on the server, and /mnt/documents with the path on your local server where you want to mount the share. From this point forward, you should be able to access the contents of the documents export on your file server. Notice, however, that the exported directory on the server was /exports/documents, but we only asked for /documents instead of the full path with the example mount command. The reason this works is because we identified an export root of /exports. To save you from flipping back, here's the first line from the /etc/exports file, where we identified our export root:

```
/exports *(ro,fsid=0,no_subtree_check)
```

With the export root, we basically set the base directory for our NFS exports. We set it as read-only (ro), because we don't want anyone making any changes to the /exports directory itself. Other directories within /exports have their own permissions and will thus override the ro setting on a per-export basis, so there's no real reason to set our export root as anything other than read-only. With our export root set, we don't have to call out the entire path of the export when we mount it; we only need the directory name. This is why we can mount an NFS export from myserver:/documents instead of having to type the entire path. While this does save us a bit of typing, it's also useful because from the user's perspective, they aren't required to know anything about the underlying filesystem on the server. There's simply no value in the user having to memorize the fact that the server is sharing a document's directory from /exports; all they're interested in is getting to their data. Another benefit is if we ever need to move our export root to a different directory (during a maintenance period), our users won't have to change their configuration to reference the new place; they'll only need to unmount and remount the exports.

So, at this point, you'll have three directories being exported from your file server, and you can always add others as you go. However, whenever you add a new export, it won't be automatically added and read by NFS. You can restart NFS to activate new exports, but that's not really a good idea while users may be connected to them, since that will disrupt their access. Thankfully, the following command will cause NFS to reread the /etc/exports file without disrupting existing connections. This will allow you to activate new exports immediately without having to wait for users to finish what they're working on:

```
sudo exportfs -a
```

With this section out of the way, you should be able to export a directory from your Ubuntu Server, and then mount that export on another Linux machine. Feel free to practice creating and mounting exports until you get the hang of it. In addition, you should familiarize yourself with a few additional options and settings that are allowable in the /etc/exports file, after consulting the man page on exports.



The `/etc/exports` file, as you've seen in this section, is typically all you'll need to set up a basic file server via NFS. However, you should also be aware that there's another configuration file, `/etc/nfs.conf`, and that file is new in Ubuntu 22.04. The purpose of this file is to tune the finer details of the NFS server. Although it's out of scope to go over the `/etc/nfs.conf` file, it's worth having a look at this file in order to get a feel for which types of options are available. For more information, see the further reading section at the end of this chapter.

When you've had more NFS practice than you can tolerate, we'll move on to a few ways in which you can copy files from one node to another without needing to set up an intermediary service or daemon.

Transferring files with `rsync`

Of all the countless tools and utilities available in the Linux and Unix world, few are as beloved as `rsync`. This is a utility that you can use to copy data from one place to another very easily, and there are many options available to allow you to be very specific about how you want the data to be transferred. Examples of its many use cases include copying files while preserving permissions, copying files while backing up replaced files, and even setting up incremental backups. If you don't already know how to use `rsync`, you'll probably want to get lots of practice with it, as it's something you'll soon see will be indispensable during your career as a Linux administrator, and it is also something that the Linux community generally assumes you already know. Thankfully, `rsync` is not hard to learn. Most administrators can learn the basic usage in about an hour or less, but the countless options available will lead you to learn new tricks even years down the road.

Another aspect that makes `rsync` flexible is the many ways you can manipulate the source and target directories. I mentioned earlier that `rsync` is a tool you can use to copy data from one place to another. The beauty of this is that the source and target can literally be anywhere you'd like, assuming you have appropriate access. For example, the most common usage of `rsync` is to copy data from a directory on one server to a directory on another server over the network. However, you don't even have to use the network; you can even copy data from one directory to another on the same server. While this may not seem like a useful thing to do at first, consider that the target directory may be a mount point that leads to a backup disk, or an NFS share that actually exists on another server. This also works in reverse: you can copy data from a network location to a local directory if you desire.

To get started with practicing with `rsync`, I recommend that you find some sample files to work with. Perhaps you have a collection of documents you can use, MP3 files, videos, text files, or basically any kind of data you have lying around. It's important to make a copy of this data. If we make a mistake we could overwrite things, so it's best to work with a copy of the data, or data you don't care about, while you're practicing. If you don't have any files to work with, you can create some text files. The idea is to practice copying files from one place to another; it really doesn't matter what you copy or where you send it to. I'll walk you through some `rsync` examples that will progressively increase in complexity. The first few examples will show you how to back up a home directory, but later examples will be potentially destructive so you will probably want to work with sample files until you get the hang of it.

Here's our first example:

```
sudo rsync -r /home/myuser /backup
```

With that command, we're using `rsync` (as root) to copy the contents of the home directory for the `myuser` directory to a backup directory, `/backup` (make sure the target directory exists). In the example, I used the `-r` option, which means `rsync` will grab directories recursively as well. You should now see a copy of the `/home/myuser` directory inside your `/backup` directory.

However, we have a bit of a problem. If you look at the permissions in the `/backup/myuser` directory, you can see that everything in the target is now owned by root. This isn't a good thing; when you back up a user's home directory, you'll want to retain their permissions. In addition, you should retain as much metadata as you can, including things like timestamps. Let's try another variation of `rsync`. Don't worry about the fact that `/backup` already has a copy of the `myuser` home directory from our previous backup. Let's perform the backup again, but this time, we'll use the `-a` option:

```
sudo rsync -a /home/myuser /backup
```

This time, we replaced the `-r` option with `-a` (archive), which retains as much metadata as possible (in most cases, it should make everything an exact copy). What you should notice now is that the permissions within the backup match the permissions within the user's home directory we copied from. The timestamps of the files will now match as well. This works because whenever `rsync` runs, it will copy what's different from the last time it ran. The files from our first backup were already there, but the permissions were wrong. When we ran the second command, `rsync` only needed to copy what was different, so it applied the correct permissions to the files. If any new files were added to the source directory since we last ran the command, the new or updated files would be copied over as well.

The archive mode (the `-a` option that we used with the previous command) is actually very popular; you'll probably see it a lot during your travels. The `-a` option is actually a wrapper option that includes the following options all at the same time:

```
-rlptgoD
```

If you're curious about what each of these options do, consult the man page for `rsync` for more detailed information. In summary, the `-r` option copies data recursively (which we already know), the `-l` option copies symbolic links, `-p` preserves permissions, `-g` preserves group ownership, `-o` preserves the owner, and `-D` preserves device files. If you put those options together, we get `-rlptgoD`. Therefore, `-a` is actually equal to `-rlptgoD`. I find `-a` easier to remember.

The archive mode is great and all, but wouldn't it be nice to be able to watch what `rsync` is up to when it runs? Add the `-v` option and try the command again:

```
sudo rsync -av /home/myuser /backup
```

This time, `rsync` will display on your terminal what it's doing as it runs (`-v` activates **verbose** mode). This is actually one of my favorite variations of the `rsync` command, as I like to copy everything and retain all the metadata, as well as watch what `rsync` is doing as it works.

What if I told you that `rsync` supports SSH by default? It's true! Using `rsync`, you can easily copy data from one node to another, even over SSH. The same options apply, so you don't actually have to do anything different other than point `rsync` to the other server, rather than to another directory on your local server:

```
sudo rsync -av /home/myuser admin@192.168.1.5:/backup
```

With this example, I'm copying the home directory for `myuser` to the `/backup` directory on server `192.168.1.5`. I'm connecting to the other server as the `admin` user. Make sure you change the user account and IP address accordingly, and also make sure the user account you use has access to the `/backup` directory. When you run this command, you should get prompted for the SSH password as you would when using plain SSH to connect to the server. After the connection is established, the files will be copied to the target server and directory.

Now, we'll get into some even cooler examples (some of which are potentially destructive), and we probably won't want to work with an actual home directory for these, unless it's a test account and you don't care about its contents. As I've mentioned before, you should have some test files to play with. When practicing, simply replace my directories with yours. Here's another variation worth trying:

```
sudo rsync -av --delete /src /target
```

Now I'm introducing you to the `--delete` option. This option allows you to synchronize two directories. Let me explain why this is important. With every `rsync` example up until now, we've been copying files from point A to point B, but we weren't deleting anything. For example, let's say you've already used `rsync` to copy contents from point A to point B. Then, you delete some files from point A. When you use `rsync` to copy files from point A to point B again, the files you deleted in point A won't be deleted in point B. They'll still be there. This is because by default, `rsync` copies data between two locations, but it doesn't remove anything. With the `--delete` option, you're effectively synchronizing the two points, thus you're telling `rsync` to make them the same by allowing it to delete files in the target that are no longer in the source.

Next, we'll add the `-b` (backup) option:

```
sudo rsync -avb --delete /src /target
```

This one is particularly useful. Normally, when a file is updated on `/src` and then copied over to `/target`, the copy on `/target` is overwritten with the new version. But what if you don't want any files to be replaced? The `-b` option renames files on the target that are being overwritten, so you'll still have the original file. If you add the `--backup-dir` option, things get really interesting:

```
sudo rsync -avb --delete --backup-dir=/backup/incremental /src /target
```

Now, we're copying files from `/src` to `/target` as we were before, but we're now sending replaced files to the `/backup/incremental` directory. This means that when a file is going to be replaced on the target, the original file will be copied to `/backup/incremental`. This works because we used the `-b` option (backup) but we also used the `--backup-dir` option, which means that the replaced files won't be renamed; they'll simply be moved to the designated directory. This allows us to effectively perform incremental backups.

Building on our previous example, we can use the Bash shell itself to make incremental backups work even better. Consider these commands:

```
CURDATE=$(date +%m-%d-%Y)
```

```
sudo rsync -avb --delete --backup-dir=/backup/incremental/$CURDATE /src /target
```

With this example, we grab the current date and store it in a variable (CURDATE). In the rsync portion of the command, we use that variable for the --backup-dir option. This will copy the replaced files to a backup directory named after the date the command was run. Basically, if today's date was 08-17-2022, the resulting command would be the same as if we had run the following:

```
sudo rsync -avb --delete --backup-dir=/backup/incremental/08-17-2022 /src /target
```

Hopefully, you can see how flexible rsync is and how it can be used to not only copy files between directories and/or nodes, but also to serve as a backup solution as well (assuming you have a remote destination to copy files to). The best part is that this is only the beginning. If you consult the man page for rsync, you'll see that there are a lot of options you can use to customize it even further. Give it some practice, and you should get the hang of it in no time.

Transferring files with SCP

A useful alternative to rsync is the **Secure Copy (SCP)** utility, which comes bundled with the OpenSSH client. It allows you to quickly copy files from one node to another. While rsync also allows you to copy files to other network nodes via SSH, SCP is more practical for one-off tasks; rsync is geared toward more complex jobs. If your goal is to send a single file or a small number of files to another machine, SCP is a great tool you can use to get the job done. If nothing else, it's yet another item for your administration toolbox. To utilize SCP, we'll use the scp command. Since you most likely already have the OpenSSH client installed, you should already have the scp command available. If you execute which scp, you should receive the following output:

```
/usr/bin/scp
```

If you don't see any output, make sure that the openssh-client package is installed.

Using SCP is very similar in nature to rsync. The command requires a source, a target, and a filename. To transfer a single file from your local machine to another, the resulting command would look similar to the following:

```
scp myfile.txt jdoe@192.168.1.50:/home/jdoe
```

With this example, we're copying the myfile.txt file (which is located in our current working directory) to a server located at 192.168.1.50. If the target server is recognized by DNS, we could've used the DNS name instead of the IP address. The command will connect to the server as user jdoe and place the file into that user's home directory. Actually, we can shorten that command a bit:

```
scp myfile.txt jdoe@192.168.1.50:
```


Notice that I removed the target path, which was `/home/jdoe`. I'm able to omit the path to the target, since the home directory is assumed if you don't give the `scp` command a target path. Therefore, the `myfile.txt` file will end up in `/home/jdoe` whether or not I included the path to the home directory explicitly. If I wanted to copy the file somewhere else, I would definitely need to call out the location. Make sure you always include at least the colon when copying a file, since if you don't include it, you'll end up copying the file to your current working directory instead of the target.

The `scp` command also works in reverse:

```
scp jdoe@192.168.1.50:myfile.txt .
```

With this example, we're assuming that `myfile.txt` is located in the home directory for the user `jdoe`. This command will copy that file to the current working directory of our local machine, since I designated the local path as a *single period* (which corresponds to our current working directory). Using `scp` in reverse isn't always practical, since you have to already know where the desired file is stored on the target before transferring it.

With our previous `scp` examples, we've only been copying a single file. If we want to transfer or download an entire directory and its contents, we will need to use the `-r` option, which allows us to do a recursive copy:

```
scp -r /home/jdoe/downloads/linux_iso jdoe@192.168.1.50:downloads
```

With this example, we're copying the local folder `/home/jdoe/downloads/linux_iso` to remote machine `192.168.1.50`. Since we used the `-r` option, `scp` will transfer the `linux_iso` folder and all of its contents. At the remote end, we're again connecting via the user `jdoe`. Notice that the target path is simply `downloads`. Since `scp` defaults to the user's home directory, this will copy the `linux_iso` directory from the source machine to the target machine under the `/home/jdoe/downloads` directory. The following command would've had the exact same result:

```
scp -r /home/jdoe/downloads/linux_iso jdoe@192.168.1.50:/home/jdoe/downloads
```

The home directory is not the only assumption the `scp` command makes. It also assumes that SSH is listening on port 22 on the remote machine. Since it's possible to change the SSH port on a server to something else, port 22 may or may not be what's in use. If you need to designate a different port for `scp` to use, use the `-P` option:

```
scp -P 2222 -r /home/jdoe/downloads/linux_iso jdoe@192.168.1.50:downloads
```

With that example, we're connecting to the remote machine via port 2222. If you've configured SSH to listen on a different port, change the number accordingly.

Although port 22 is always the default for OpenSSH, it's common for some administrators to change it to something else. While changing the SSH port doesn't add a great deal of benefit in regard to security (an intensive port scan will still find your SSH daemon), it's a relatively easy change to make, and making it even just a little bit harder to find is beneficial. We'll discuss this further in *Chapter 21, Securing Your Server*.

Like most commands in the Linux world, the `scp` command supports verbose mode. If you want to see how the `scp` command progresses as it copies multiple files, add the `-v` option:

```
scp -rv /home/jdoe/downloads/linux_iso jdoe@192.168.1.50:downloads
```

Well, there you have it. The `scp` command isn't overly complex or advanced, but it's really great for situations in which you want to perform a one-time copy of a file from one node to another. Since it copies files over SSH, you benefit from its security, and it also integrates well with your existing SSH configuration. An example of this integration is the fact that `scp` recognizes your `~/.ssh/config` file (if you have one), so you can shorten the command even further. Go ahead and practice with it a bit, and in the next section, we'll go over yet another trick that OpenSSH has up its sleeve.

Summary

In this chapter, we explored multiple ways of accessing remote resources. Just about every network has a central location for storing files, and we explored two ways of accomplishing this with NFS and Samba. Both NFS and Samba have their place in the data center and are very useful ways in which we can make resources on a server available to our users who need to access them. We also talked about `rsync` and `scp`, two great utilities for transferring data without needing to set up a permanent share.

Next up is *Chapter 13, Managing Databases*. Now that we have all kinds of useful services running on our Ubuntu Server network, it's only fitting that we take a look at serving databases as well. Specifically, we'll look at MariaDB. See you there!

Relevant videos

- Tutorial for `scp`: <https://linux.video/scp>
- Backing up a Linux server with `rsync`: <https://linux.video/rsync-backup>
- Transferring files with `rsync`: <https://linux.video/rsync-transfer>

Further reading

- Ubuntu `rsync` documentation: <https://learnlinux.link/u-rsync>
- Ubuntu Samba documentation: <https://learnlinux.link/u-smb>
- Service – NFS (contains additional information on NFS changes in 22.04): <https://learnlinux.link/u-nfs-1>
- Ubuntu NFS documentation: <https://learnlinux.link/u-nfs-2>
- Active Directory Integration: <https://learnlinux.link/u-ad>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



13

Managing Databases

The Linux platform has long been a very popular choice for hosting databases. Given the fact that databases power a large majority of popular websites across the internet nowadays, this is a very important role for servers to fill. Ubuntu Server is also a very popular choice for this purpose, as its stability is a major benefit to the hosting community. This time around, we'll take a look at MariaDB, a popular fork of MySQL. The goal won't be to provide a full walkthrough of MySQL's syntax (as that would be a full book in and of itself), but we'll focus on setting up and maintaining database servers utilizing MariaDB, and we'll even go over how to set up a primary/secondary relationship between them. If you already have a firm understanding of how to architect databases, you'll still benefit from this chapter as we'll be discussing Ubuntu's implementation of MariaDB in particular, which has its configuration organized a bit differently than in other platforms.

As we work through setting up our very own MariaDB server, we will cover the following topics:

- Preparations for setting up a database server
- Installing MariaDB
- Understanding the MariaDB configuration files
- Managing MariaDB databases
- Setting up a secondary database server

As with setting up any server for a particular purpose, we should first think about the goal and plan accordingly. So in the next section, we'll talk about some of the higher-level details to consider while preparing a database server.

Preparations for setting up a database server

Before we get started with setting up our database server, there are a few odds and ends to get out of the way. As we go through this chapter, we'll set up a basic database server using MariaDB. I'm sure more than a few of my readers are familiar with MySQL. MySQL is a tried and true solution that is still in use in many data centers today, and that will probably continue to be the case for the foreseeable future. There's a good chance that a popular website or two that you regularly visit utilizes it on the backend. So, you may be wondering then, why not go over that instead of MariaDB?

There are two reasons why this book will focus on MariaDB. First, the majority of the Linux community is migrating over to it (more on that later), and it's also a drop-in replacement for MySQL. This means that any databases or scripts you've already written for MySQL will most likely work just fine with MariaDB, barring some edge cases. In reverse, the commands you practice with MariaDB should also function as you would expect on a MySQL server. This is great, considering that many MySQL installations are still in use in many data centers, and you'll be able to support those too. For the most part, there are very few reasons to stick with MySQL when your existing infrastructure can be ported over to MariaDB, and that's the direction the Linux community is headed toward anyway.

Why the change? A while back, various distributions of Linux started switching to MariaDB from MySQL. Red Hat is one such example; it switched to MariaDB back in version 7 of Red Hat Enterprise Linux. Other distributions, such as Arch Linux and Fedora, went down the same route. This was partly due to a lack of trust in Oracle, the company that now owns MySQL. When Oracle became the owner of MySQL, there were some serious questions raised in the open source community regarding the future of MySQL as well as its licensing. I'm not going to get into any speculation about Oracle, the future of MySQL, or any politics regarding its future since it's not relevant to this book (and I'm not a fan of corporate drama). The fact, though, is that many distributions are moving toward MariaDB, and that seems to be the future. It's a great technology, and I definitely recommend it over MySQL for several reasons that are unrelated to current controversies.

MariaDB is more than just a fork of MySQL. On its own, it's a very competent database server. The fact that your existing MySQL implementations should be compatible with it eases adoption. But more than that, MariaDB makes some very worthwhile changes and improvements to MySQL that will only benefit you.

Everything you love about MySQL can be found in MariaDB, plus some cutting-edge features that are exclusive to it. But even better is the fact that MariaDB features additional clustering options that are leaps and bounds better and more efficient than plain old MySQL.

So, hopefully I've sold you on the value of MariaDB. Ultimately, whether or not you actually use it will depend on the needs of your organization. I've seen some organizations opt to stick with MySQL, if only for the sole reason that it's what they know. I can understand that if a solution has proven itself in your data center, there's really no reason to change if your database stack is working perfectly fine the way it is. To that end, while I'll be going over utilizing MariaDB, it's possible that the examples may work for MySQL as well. If in doubt, MariaDB is recommended for the examples in this chapter.

With regard to your server, a good implementation plan is key (as always). I won't spend too much time on this aspect, since by now I know you've probably been through a paragraph or two in this book where I've mentioned the importance of redundancy (and I'm sure I'll mention redundancy again a few more times before the last page). At this point, you're probably just setting up a lab environment or test server on which to practice these concepts before using your newfound skills in production. But when you do eventually roll out a database server into production, it's crucial to plan for long-term stability. Database servers should be regularly backed up, redundant (there I go again), and regularly patched. Later on in this chapter, I'll walk you through setting up a secondary database server, which will take care of the redundancy part.

However, that's not enough on its own, as regular backups are important. There are many utilities that allow you to do this, such as `mysqldump`, and you can also take snapshots of your virtual machine (assuming you're not using a physical server). Both solutions are valid, depending on your environment. As someone who has lost an entire workday attempting to resurrect a fallen database server for a client (of which they had no backups or redundancy), my goal is simply to spare you that headache.

As far as how many resources a database server needs, that solely depends on your environment. MariaDB itself does not take up a huge amount of resources, but as with MySQL, your usage is dependent on your workload. Either you'll have a few dozen clients connecting, or a few thousand, or more. But one recommendation I'll definitely make is to use **Logical Volume Manager (LVM)** for the partition that houses your database files. This will certainly spare you grief in the long run. As we've discussed in *Chapter 9, Managing Storage Volumes*, LVM makes it very simple to expand a filesystem, especially on a virtual machine.

If your database server is on a virtual machine, you can add a disk to the volume group and expand it if your database partition starts to get full, and your customers will never notice there was ever about to be a problem. Without LVM, you'll need to shut down the server, add a new volume, `rsync` your database server files over to the new location, and then bring up the server. Depending on the size of your database, this situation can span hours. Do yourself a favor, use LVM.

With that out of the way, we can begin setting up MariaDB. For learning and testing purposes, you can use pretty much any server you'd like: physical, virtual, or a **Virtual Private Server (VPS)**. Once you're ready, let's move on and we'll get started!

Installing MariaDB

Now we've come to the fun part, installing MariaDB. To get the ball rolling, we'll install the `mariadb-server` package:

```
sudo apt install mariadb-server
```

If your organization prefers to stick with MySQL, the package to install is `mysql-server` instead:

```
sudo apt install mysql-server
```

Although it might be tempting to try out both MySQL and MariaDB to compare and contrast their differences, I don't recommend switching from MariaDB to MySQL (or vice versa) on the same server. I've seen some very strange configuration issues occur on servers that had one installed and then were switched to the other (even after wiping the configuration). For the most part, it's best to pick one solution per server and stick with it. As a general rule, MySQL should be used if you have legacy databases to support. For brand-new installations, go with MariaDB.

After you install the `mariadb-server` package, check to make sure the service started and is enabled. By default, it should already be running:

```
systemctl status mariadb
```

Next, we'll want to add some security to our MariaDB installation (even though we're using MariaDB, the name of the following command hasn't been updated and still includes `mysql` in the name):

```
sudo mysql_secure_installation
```

At this point, we haven't set a root password yet, so go ahead and just press *Enter* when the script asks for it. This script will ask you additional questions. You will be asked if you'd like to switch to **unix_socket authentication**. The default answer for this question is yes, but I recommend answering no to this question. This will ensure password authentication is used, which is going to be better for compatibility. There is a security benefit by switching to `unix_socket` authentication, but for our purposes password authentication will suffice. If you decide to dive into the subject of database management, switching to `unix_socket` authentication might be better in the future.

Continuing, the `mysql_secure_installation` script will ask you if you want to set a root password. The root user for MariaDB is not the same as the root user on your system, and you definitely should create a password for it. So when this comes up, press *y* to tell it you want to create a root password, then enter that password twice.

After setting the root password, the script will ask you whether you'd like to remove anonymous users, and also disallow remote access to the database server. You should answer yes to both. The latter is especially important, as there's almost never a situation in which allowing public access to MySQL/MariaDB is a good idea. Even if you're hosting a website for external users, those users only need access to the website, not the database server. The website itself will interface with the database locally as needed; an external connection wouldn't be necessary. (If you have a standalone database server that's not publicly accessible to the internet, then you will want to answer yes here – just make sure that the database server can't be reached publicly). Continuing for the most part, just answer yes to everything the script asks you (unless you have a good reason not to).

The entire process after executing `mysql_secure_installation` looks like the following, and I've included the responses I recommend for each (“y” for yes, “n” for no):

```
Enter current password for root (enter for none):  
Switch to unix_socket authentication [Y/n] n  
Set root password? [Y/n] y  
Remove anonymous users? [Y/n] y  
Disallow root login remotely? [Y/n] y  
Remove test database and access to it? [Y/n] y  
Reload privilege tables now? [Y/n] y
```

At this point, we officially have a fully functional database server. The previous command allowed us to apply some basic security, and our database server is now available to us. To connect to it and manage it, we'll use the `mariadb` command to access the MariaDB shell, where we'll enter commands to manage our database(s). There are actually two methods to connect to this shell. The first method is by simply using the `mariadb` command with `sudo`:

```
sudo mariadb
```

This particular command works because if you use the `mariadb` command as `root` (we used `sudo` in this example) the password is bypassed. In fact, we didn't even enter the username either; `root` is assumed if you are attempting to access MariaDB with `sudo`. This is by far the simplest way to connect. However, some of you may be accustomed to a different method of authentication if you've used other Linux distributions: entering the username and password. In that case, the command will look like this (it won't work by default though):

```
mariadb -u root -p
```

When that command works correctly, it will ask you for your root password and then let you into the shell.

It's recommended to create a different user in order to manage your MariaDB installation, as logging in as `root` is not recommended in most cases. We'll be creating additional users later on in this chapter, but for now, the `root` account is the only one we have available. It's common practice to use the `root` account to do the initial setup, and then create a different user for administrative purposes going forward. However, the `root` account is still often used for server maintenance, so use your best judgment.

So, now that we have access to the MariaDB shell, what can we do with it? The commands we'll execute on this shell allow us to do things such as create and delete databases and users, add tables, and so on. The `mariadb` command comes from the `mariadb-client-10.6` package, which was installed as a dependency when we installed `mariadb-server`. Entering the `mariadb` command by itself with no options connects us to the database server on our local machine. This utility also lets us connect to external database servers to manage them remotely, which we'll discuss later.

The MariaDB shell prompt will look like this:

```
MariaDB [(none)]>
```

We'll get into MariaDB commands and user management later. For now, you can exit the shell. To exit, you can type `exit` and press *Enter* or press *Ctrl + d* on your keyboard.

Now, our MariaDB server is ready to go. While you can now move on to the next section, you might want to consider setting up another MariaDB server by following these steps on another machine. If you have room for another virtual machine, it might be a good idea for you to get this out of the way now, since we'll be setting up a secondary database server later.

Understanding the MariaDB configuration files

Now that we have MariaDB installed, let's take a quick look at how its configuration is stored. While we won't be changing much of the configuration in this chapter (aside from adding parameters related to setting up a secondary database instance), it's a good idea to know where to find the configuration, since you'll likely be asked by a developer to tune the database configuration at some point in your career. This may involve changing the storage engine, buffer sizes, or countless other settings. A full walkthrough on performance tuning is outside the scope of this book, but it will be helpful to know how the settings for MariaDB are read, since Ubuntu's implementation is fairly unique.

The configuration files for MariaDB are stored in the `/etc/mysql` directory. In that directory, you'll see the following files by default:

```
debian.cnf
debian-start
mariadb.cnf
my.cnf
my.cnf.fallback
```

You'll also see the following directories:

```
conf.d
mariadb.conf.d
```

The configuration file that MariaDB reads on startup is the `/etc/mysql/mariadb.cnf` file. This is where you'll begin perusing when you want to configure the daemon, but we'll get to that soon. The `/etc/mysql/debian-start` file is actually a script that sets default values for MariaDB when it starts, such as setting some environment variables. It also defines a task that is executed if the `mariadb` process dies or exits, and allows it to check for crashed tables.

The `debian-start` script also loads the `/etc/mysql/debian.cnf` file, which sets some client settings for the `mariadb` daemon. Here's a list of values from that file:

```
[client]
host      = localhost
user      = root
[mysql_upgrade]
host      = localhost
user      = root
```

The defaults for these values are fine and there's rarely a reason to change them. Essentially, the file sets the default user, host, and socket location. If you've used MySQL before on other platforms, you may have seen many of those settings in the `/etc/my.cnf` file, which is typically the standard file for the `mariadb` daemon. With MariaDB on Ubuntu Server, you can see that the default layout of the files was changed considerably.

The `/etc/mysql/mariadb.cnf` file sets the global defaults for MariaDB. However, in Ubuntu's implementation, this default file just includes configuration files from the `/etc/mysql/conf.d` and `/etc/mysql/mariadb.conf.d` directories. Within those directories, there are additional files ending with the `.cnf` extension. Many of these files contain default configuration values that would normally be found in a single file, but Ubuntu's implementation modularizes these settings into separate files instead. For our purposes in this book, we'll be editing the `/etc/mysql/conf.d/mysql.cnf` file when it is time to set up a relationship between primary and secondary servers.

The other configuration files aren't relevant for the content of this book, and their current values are more than sufficient for what we need. When it comes to performance tuning, you may consider creating a new configuration file ending with the `.cnf` extension, with specific tuning values as provided by the documentation of a software package you want to run that interfaces with a database, or requirements given to you by a developer.

For additional information on how these configuration files are read, you can refer to the `/etc/mysql/mariadb.cnf` file, which includes some helpful content at the top of the file that details the order in which these configuration files are read, as well as their purpose. Here's an excerpt of these comments from that file:

```
# The MariaDB/MySQL tools read configuration files in the following order:
# 1. "/etc/mysql/mariadb.cnf" (this file) to set global defaults,
# 2. "/etc/mysql/conf.d/*.cnf" to set global options.
# 3. "/etc/mysql/mariadb.conf.d/*.cnf" to set MariaDB-only options.
# 4. "~/.my.cnf" to set user-specific options.
```

As we can see, when MariaDB starts up, it first reads the `/etc/mysql/mariadb.cnf` file, followed by `.cnf` files stored within the `/etc/mysql/conf.d` directory, then the `.cnf` files stored within the `/etc/mysql/mariadb.conf.d` directory, followed by any user-specific settings stored within a `.my.cnf` file that may be present in the user's home directory.

With Ubuntu's implementation, when the `/etc/mysql/mariadb.cnf` file is read during startup, the process will immediately scan the contents of `/etc/mysql/conf.d` and `/etc/mysql/mariadb.conf.d`, because the `/etc/mysql/mariadb.cnf` file contains the following lines:

```
!includedir /etc/mysql/conf.d/
!includedir /etc/mysql/mariadb.conf.d/
```

As you can see, the order that the configuration files are checked is set to the `mariadb.cnf` file first, followed by the `/etc/mysql/conf.d` and `/etc/mysql/mariadb.conf.d` directories.

This may be a bit confusing at first, because the default configuration for MariaDB in Ubuntu Server essentially consists of files that redirect to other files. But the main takeaway is that any configuration changes you make that are not exclusive to MariaDB (basically, configuration that's compatible with MySQL itself) should be placed in a configuration file that ends with the `.cnf` extension, and then stored in the `/etc/mysql/conf.d` directory.

If the configuration you're wanting to add is for a feature exclusive to MariaDB (but not compatible with MySQL itself), the configuration file should be placed in the `/etc/mysql/mariadb.conf.d` directory instead. For our purposes, we'll be editing the `/etc/mysql/conf.d/mysql.cnf` file when it comes time to set up our primary/secondary replication, since the method we'll be using is not specific to MariaDB.

In this section, we discussed MariaDB configuration and how it differs from its implementation in other platforms. The way the configuration files are presented is not the only difference in Ubuntu's implementation of MariaDB; there are other differences as well. In the next section, we'll take a look at a few additional ways in which Ubuntu's implementation differs from implementations in other distributions of Linux.

Managing MariaDB databases

Now that our MariaDB server is up and running, we can finally look into managing it. In this section, I'll demonstrate how to connect to a database server using the `mariadb` command, which will allow us to create databases, remove (drop) them, and also manage users and permissions.

To begin, we'll need to create an administrative user for MariaDB. The root account already exists as the default administrative user, but it's not a good idea to allow others to use that account. Instead, it makes more sense to create an administrative account separate from root for managing our databases. Therefore, we'll begin our discussion on managing databases with user management. The users we'll manage within MariaDB are specific to MariaDB; these are separate from the user accounts on the actual system.

To manage and interact with databases, we'll need to enter the MariaDB shell, and the same goes for when it comes to creating database users. Right now, since we only have the root account, we'll need to access the current MariaDB implementation as root in order to set up our administrative user. If you've set up standard authentication, as we discussed earlier in this chapter, you can access the prompt using the standard means:

```
mariadb -u root -p
```

Or, more simply:

```
sudo mariadb
```

Once inside the MariaDB shell, your prompt will change to the following:

```
MariaDB [(none)]>
```

Now, we can create our new administrative user. I'll call mine `admin` in my examples, but you can use whatever name you'd like. In a company I used to work for, we used the username `velociraptor` as our administrative user on our servers, since nothing is more powerful than a `velociraptor` (and they can open doors). Feel free to use a clever name, but just make sure you remember it. Using a non-standard username has the added benefit of security by obscurity; the name wouldn't be what an intruder would expect.

Here's the command to create a new user in MariaDB (replace the username and password in the command with your desired credentials):

```
CREATE USER 'admin'@'localhost' IDENTIFIED BY 'password';  
FLUSH PRIVILEGES;
```

When it comes to MySQL syntax, the commands are not case sensitive (though the data parameters are), but it's common to capitalize instructions to separate them from data. During the remainder of this chapter, we'll be executing some commands within the Linux shell, and others within the MariaDB shell. I'll let you know which shell each command needs to be executed in as we come to them, but if you are confused, just keep in mind that MariaDB commands are the only ones that are capitalized.

With the preceding commands, we're creating the `admin` user and restricting it to `localhost`. This is important because we don't want to open up the `admin` account to the world. We're also flushing privileges, which causes MariaDB to reload its privileges information. The `FLUSH PRIVILEGES` command should be run every time you add a user or modify permissions. I may not always mention the need to run this command, so you might want to make a mental note of it and make it a habit now.

As I mentioned, the previous command created the `admin` user but is only allowing it to connect from `localhost`. This means that an administrator would first need to log in to the server itself before they would be able to log in to MariaDB with the `admin` account. As an example of the same command (but allowing remote login from any other location), the following command is a variation that will do just that:

```
CREATE USER 'admin'@'%' IDENTIFIED BY 'password';
```

Can you see the percent symbol (%) in place of `localhost`? That basically means *everywhere*, which indicates we're creating a user that can be logged into from any source (even external nodes). However, by restricting our user to `localhost` with the first command, we're making our server just a bit more secure. You can also restrict access to particular networks, which is desired if you really do need to allow a database administrator access to the server remotely:

```
CREATE USER 'admin'@'192.168.1.%' IDENTIFIED BY 'password';
```

That's a little better, but not as secure as limiting login to `localhost`. As you can see, the % character is basically a wildcard, so you can restrict access to needing to be from a specific IP or even a particular subnet.

So far, all we did is create a new user; we have yet to give this user any permissions. We can create a set of permissions (also known as **grants**) with the `GRANT` command. First, let's give our `admin` user full access to the database server when called from `localhost`:

```
GRANT ALL PRIVILEGES ON *.* TO 'admin'@'localhost';  
FLUSH PRIVILEGES;
```

Now, we have an administrative user we can use to manage our server's databases. We can use this account for managing our server instead of the `root` account. Any logged-on Linux user will be able to access the database server and manage it, provided they know the password. To access the MariaDB shell as the `admin` user that we created, the following command will do the trick:

```
mariadb -u admin -p
```

After entering the password, you'll be logged into MariaDB as `admin`.

In addition, you can actually provide the password to the `mariadb` command without needing to be prompted for it:

```
mariadb -u admin -p<password>
```

Notice that there is no space in between the `-p` option and the actual password (though it's common to put a space between the `-u` option and the username). As useful as it is to provide the username and password in one shot, I don't recommend that you ever use that method. This is because any Linux command you type is saved in the history, so anyone can view your command history and they'll see the password in plain text. I only mention it here because I find that many administrators do this, even though they shouldn't. At least now you're aware of this method and why it's wrong.

The `admin` account we created is only intended for system administrators who need to manage databases on the server. The password for this account should not be given to anyone other than staff employees or administrators that absolutely need it. Additional users can be added to the MariaDB server, each with differing levels of access. Keep in mind that our `admin` account can manage databases but not users. This is important, as you probably shouldn't allow anyone other than server administrators to create users. You'll still need to log in as `root` to manage user permissions.

It may also be useful to create a read-only user for MariaDB for employees who need to be able to read data but not make changes. Back in the MariaDB shell (as `root`), we can issue the following command to effectively create a read-only user:

```
GRANT SELECT ON *.* TO 'readonlyuser'@'localhost' IDENTIFIED BY 'password';
```

With this command (and flushing privileges afterward), we've done two things. First, we've created a new user and also set up grants for that user with a single command. Second, we've created a read-only user that can view databases but not manage them (we've restricted the permissions to `SELECT`). This is more secure. In practice, it's better to restrict a read-only user to a specific database. This is typical in a development environment, where you'll have an application that connects to a database over a network and needs to read information from it. We'll go over this scenario soon.

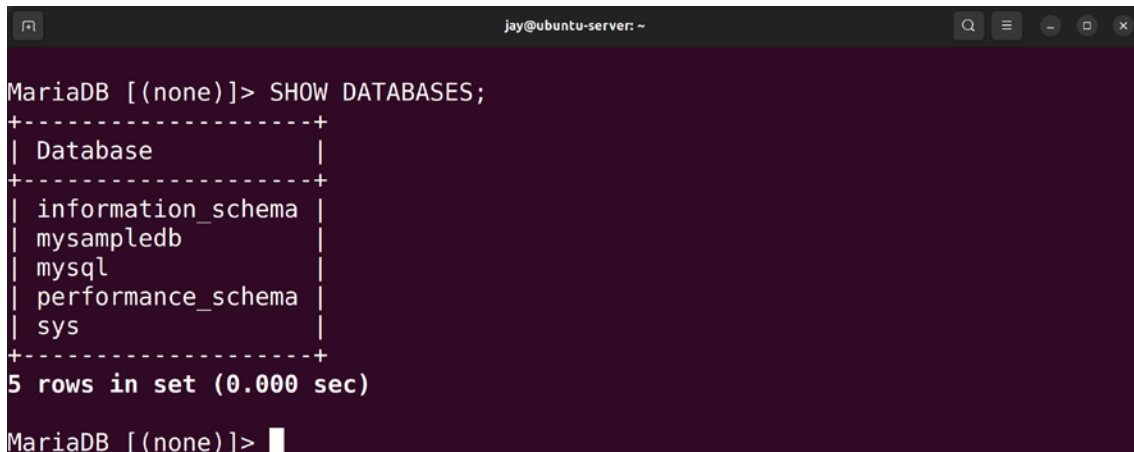
Next, let's create a database. At the MariaDB prompt, execute:

```
CREATE DATABASE mysampledb;
```

That was easy. We should now have a database on our server named `mysampledb`. To list all databases on our server (and confirm our database was created properly), we can execute the following command:

```
SHOW DATABASES;
```

This will produce an output like the following:



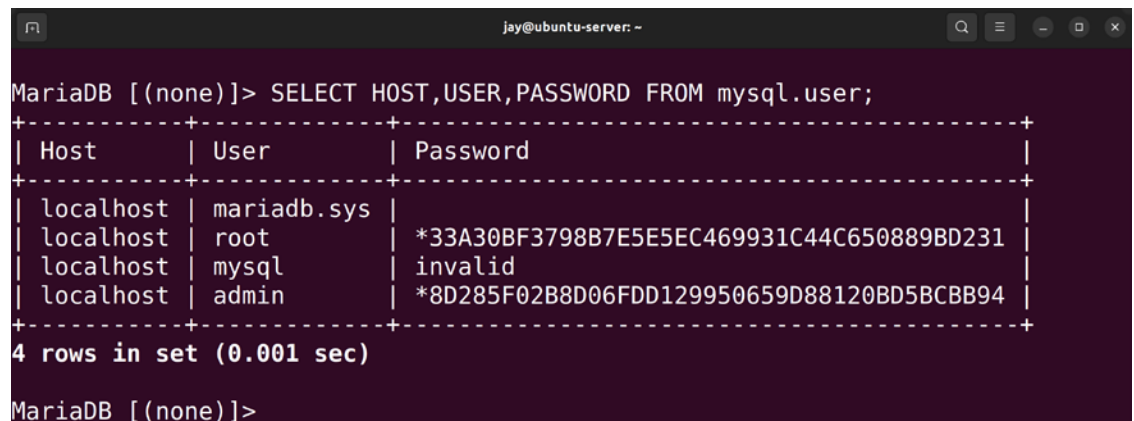
```
jay@ubuntu-server: ~  
MariaDB [(none)]> SHOW DATABASES;  
+-----+  
| Database |  
+-----+  
| information_schema |  
| mysampled |  
| mysql |  
| performance_schema |  
| sys |  
+-----+  
5 rows in set (0.000 sec)  
MariaDB [(none)]>
```

Figure 13.1: Listing MariaDB databases

The output will show some system databases that were created for us, but our new database should be listed among them. We can also list users just as easily:

```
SELECT HOST, USER, PASSWORD FROM mysql.user;
```

Entering this command will result in something similar to the following output:



```
jay@ubuntu-server: ~  
MariaDB [(none)]> SELECT HOST, USER, PASSWORD FROM mysql.user;  
+-----+-----+-----+  
| Host      | User      | Password |  
+-----+-----+-----+  
| localhost | mariadb.sys |          |  
| localhost | root      | *33A30BF3798B7E5E5EC469931C44C650889BD231 |  
| localhost | mysql     | invalid  |  
| localhost | admin     | *8D285F02B8D06FDD129950659D88120BD5BCBB94 |  
+-----+-----+-----+  
4 rows in set (0.001 sec)  
MariaDB [(none)]>
```

Figure 13.2: Listing MariaDB users

In a typical scenario, when installing an application that needs its own database, we'll create the database and then a user for that database. We'll normally want to give that user permission to only that database, with only as much permission as required to allow it to function properly. We've already created the `mysampled` database, so if we want to create a user with read-only access to it, we can do so with the following command:

```
GRANT SELECT ON mysampled.* TO 'appuser'@'localhost' IDENTIFIED BY 'password';
```

With one command, we're not only creating the user `appuser`, but we're also setting a password for it, in addition to allowing it to have `SELECT` permissions on the `mysampled` database. This is equivalent to read-only access. If our user needed full access, we could use the following instead:

```
GRANT ALL ON mysampled.* TO 'appuser'@'localhost' IDENTIFIED BY 'password';
```

To double-check that we've executed the command correctly, we can use this command to show the grants for a particular user:

```
SHOW GRANTS FOR 'appuser'@'localhost';
```

Now, our `appuser` has full access but only to the `mysampled` database. Of course, we should only provide full access to the database if absolutely necessary. We can also provide additional permissions, such as `DELETE` (whether or not the user has permission to delete rows from database tables), `CREATE` (which controls whether the user can add rows to the database), `INSERT` (controls whether or not the user can add new rows to a table), `SELECT` (allows the user to read information from the database), `DROP` (allows the user to fully remove a database), and `ALL` (which gives the user everything). There are other permissions we can grant or deny; check the MariaDB documentation for more details. The types of permissions you'll need to grant to a user to satisfy the application you're installing will depend on the documentation for that software. Always refer to the installation instructions for the application you're attempting to install to determine which permissions are required for it to run.

If you'd like to remove user access, you can use the following command to do so (substituting `myuser` with the user account you wish to remove):

```
DROP USER 'myuser'@'localhost'
```

Now, let's go back to databases. Now that we've created the `mysampled` database, what can we do with it? We'll add tables and rows, of course! A database is useless without actual data, so we can work through some examples of adding data to our database to see how this works. First, log in to the MariaDB shell as a user with full privileges to the `mysampled` database. Now, we can have some fun and modify the contents. Here are some examples you can follow:

```
USE mysampled;
```

The `USE` command allows us to select a database we want to work with. The MariaDB prompt will change from `MariaDB [(none)]>` to `MariaDB [mysampled]>`. This is very useful, as the MariaDB prompt changes to indicate which database we are currently working with. We basically just told MariaDB that for all of the commands we're about to execute, we would like them to be issued against the `mysampled` database.

Now, we can CREATE a table in our database. It doesn't matter what you call yours, since we're just practicing. I'll call mine Employees:

```
CREATE TABLE Employees (Name char(15), Age int(3), Occupation char(15));
```

We can verify this command by showing the columns in the database, to ensure it shows what we expect:

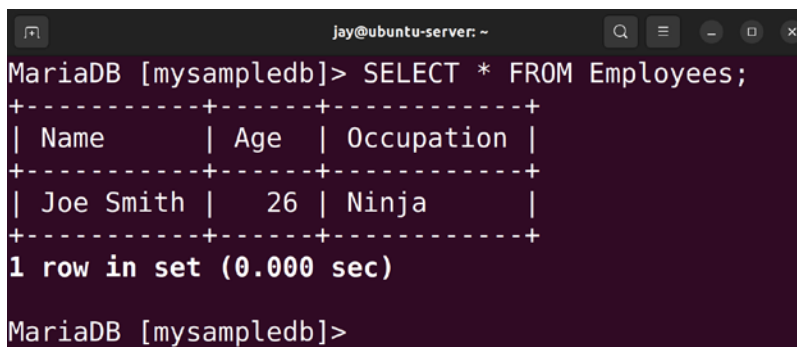
```
SHOW COLUMNS IN Employees;
```

With this command, we've created a table named Employees that has three columns (Name, Age, and Occupation). To add new data to this table, we can use the following INSERT command:

```
INSERT INTO Employees VALUES ('Joe Smith', '26', 'Ninja');
```

The example INSERT command adds a new employee to our Employees table. When we use INSERT, we insert all the data for each of the columns. Here, we have an employee named Joe, who is 26 years old and whose occupation is a Ninja. Feel free to add additional employees; all you would need to do is formulate additional INSERT statements and provide data for each of the three fields. When you're done, you can use the following command to show all of the data in this table:

```
SELECT * FROM Employees;
```



```
jay@ubuntu-server: ~
MariaDB [mysampled] > SELECT * FROM Employees;
+-----+-----+-----+
| Name      | Age  | Occupation |
+-----+-----+-----+
| Joe Smith | 26   | Ninja      |
+-----+-----+-----+
1 row in set (0.000 sec)

MariaDB [mysampled] >
```

Figure 13.3: Listing database rows from a table

To remove an entry, the following command will do what we need:

```
DELETE FROM Employees WHERE Name = 'Joe Smith';
```

Basically, we're using the DELETE FROM command, giving the name of the table we wish to delete from (Employees, in this case), and then using WHERE to provide some search criteria for narrowing down our command.

The DROP command allows us to delete tables or entire databases, and it should be used with care. I don't actually recommend you delete the database we just created, since we'll use it for additional examples. But if you really wanted to drop the Employees table, you could use:

```
DROP TABLE Employees;
```


Or use this to drop the entire database:

```
DROP DATABASE mysamp1edb;
```

There is, of course, much more to MariaDB and its MySQL syntax than the samples I have provided, but this should be enough to get you through the examples in this book. As much as I would love to give you a full walkthrough of the MySQL syntax, it would easily push this chapter beyond a reasonable number of pages. If you'd like to push your skills beyond the samples of this chapter, there are great books available that are dedicated to the subject.

Before I close this section though, I think it will be worthwhile for you to see how to back up and restore your databases. To do this, we have the `mysqldump` command at our disposal. Its syntax is very simple, as you'll see.

First, exit the MariaDB shell and return to your standard Linux shell. Since we've already created an `admin` user earlier in the chapter, we'll use that user for the purposes of our backup:

```
mysqldump -u admin -p --databases mysamp1edb > mysamp1edb.sql
```

With this example, we're using `mysqldump` to create a copy of the `mysamp1edb` database and storing it in a file named `mysamp1edb.sql`. Since MariaDB requires us to log in, we authenticate to MariaDB using the `-u` option with the username `admin` and the `-p` option, which will prompt us for a password. The `--databases` option is necessary because, by default, `mysqldump` does not include the database create statement. However, the `--databases` option forces this, which just makes it easier for you to restore. Assuming that we were able to authenticate properly, the contents of the `mysamp1edb` database will be dumped into the `mysamp1edb.sql` file. This export should happen very quickly, since this database probably only contains a single table and a few rows. Larger production databases can take hours to dump.

Restoring a backup is fairly simple. We can utilize the `mariadb` command with the backup file used as a source of input:

```
sudo mariadb -u admin -p < mysamp1edb.sql
```

So, there you have it. The `mysqldump` command is definitely very handy in backing up databases. In the next section, we'll work through setting up a secondary database server.

Setting up a secondary database server

Redundancy is an amazing thing. If a primary server fails for some reason, you can keep your applications running by having a secondary database server available in case the original meets its demise. Of course, you can always create regular backups of your database servers and restore if necessary, but it's very hard to keep up with databases that are always changing, so backups have a tendency to become stale quite fast. A secondary database server enables you to have a copy that is always up to date. This doesn't mean that you no longer need backups, but it does give you another option for recovery when faced with a problem.

The industry is moving away from terms like “Master” and “Slave” to describe a primary and secondary server. In this chapter and moving forward, we will use the terms “Primary” and “Secondary” to describe the relationship of one main database server that replicates to another. Therefore, the verbiage around our actions will use “primary” and “secondary,” even though the verbiage inside Ubuntu 22.04 still uses the old terminology. Just keep in mind that the naming will likely switch to a new naming scheme in the future.

At this point, we have one database server already. To set up a secondary database instance, all you really need in order to begin the process is to set up another physical server or virtual machine and install the `mariadb-server` package as we did earlier. If you’ve already set up two database servers as recommended earlier in the chapter, you’re ready to begin. If not, feel free to spin up another virtual machine and follow the process from the *Installing MariaDB* section, which covered the initial setup of MariaDB. Go ahead and set up another server if you haven’t already done so. Of your two servers, one should be designated as the primary and the other as the secondary, so make a note of the IP addresses for each.

To begin, we’ll first start working on the primary. We’ll need to edit the `/etc/mysql/conf.d/mysql.cnf` file on the server you wish to be the primary. Currently, the file contains just the following line:

```
[mysql]
```

Right underneath that, add a blank line and then the following code:

```
[mysqld]
log-bin
binlog-do-db=mysampled
server-id=1
```

With this configuration, we’re first enabling binary logging, which is required for a primary/secondary server to function properly. Binary logs contain records of all database changes, which enables a secondary database instance to reproduce changes made to the primary server. These binary logs record changes made to a database, which will then be transferred to a secondary server.

Another configuration file that we’ll need to edit is `/etc/mysql/mariadb.conf.d/50-server.cnf`. In this file, we have the following line:

```
bind-address = 127.0.0.1
```

With this default setting, the `mysql` daemon is only listening for connections on `localhost` (`127.0.0.1`), which is a problem since we’ll need to connect to it from another machine (the secondary server). Change this line to the following:

```
bind-address = 0.0.0.0
```

Next, we’ll need to access the MariaDB shell on the primary server and execute the following commands:

```
GRANT REPLICATION SLAVE ON *.* to 'replicate'@'192.168.1.204' identified by 'password';
```

Here, we're creating a replication user named `replicate` and allowing it to connect to our primary server from the IP address `192.168.1.204`. Be sure to change that IP to match the IP of your secondary server, but you can also use a hostname identifier such as `%.mydomain` if you have a domain configured, which is equivalent to allowing any hostname that ends with `.mydomain`. Also, we're setting the password for this user to `password`, so feel free to customize that as well to fit your password requirements (be sure to make a note of the password).

We should now restart the `mariadb` daemon so that the changes we've made to the `mysql.cnf` file take effect:

```
sudo systemctl restart mariadb
```

Next, we'll set up the secondary server. But before we do that, there's a consideration to make now that will possibly make the process easier on us. In a production environment, it's very possible that data is still being written to the primary server. The process of setting up a secondary server is much easier if we don't have to worry about the primary database changing while we set up the secondary. The following command, when executed within the MariaDB shell, will lock the database and prevent additional changes:

```
FLUSH TABLES WITH READ LOCK;
```

If you're absolutely sure that no data is going to be written to the primary server, you can disregard that step.

Next, we should utilize `mysqldump` to make sure that both the primary and the secondary servers contain the same data before we start synchronizing them. The process is smoother if we begin with them already synchronized, rather than trying to mirror the databases later. Using `mysqldump` as we did in the previous section, create a dump of the primary server's database and then import that dump into the secondary. The easiest way to transfer the dump file is to use `rsync` or `scp`. Then, on the secondary instance, use `mariadb` to import the file.

The command to back up the database on the primary server becomes the following:

```
mysqldump -u admin -p --databases mysampled > mysampled.sql
```

After transferring the `mysampled.sql` file to the secondary server, you can import the backup into the secondary server:

```
mariadb -u root -p < mysampled.sql
```

Also on the secondary server, we'll need to edit `/etc/mysql/conf.d/mysql.cnf` and then place the following code at the end (make sure to add a blank line after `[mysql]`):

```
[mysqld]
server-id=2
```

Although it's outside the scope of this book, you can set up more than just one secondary database server. If you do, each will need a unique `server-id`.

Make sure you restart the mariadb unit on the secondary server before continuing:

```
sudo systemctl restart mariadb
```

From the root MariaDB shell on your secondary server, enter the following command. Change the IP address in the command accordingly:

```
CHANGE MASTER TO MASTER_HOST="192.168.1.184", MASTER_USER='replicate', MASTER_PASSWORD='password';
```

Now that we're finished configuring the synchronization, we can unlock the primary server's tables. On the primary server, execute the following command within the MariaDB shell:

```
UNLOCK TABLES;
```

Now, we can check the status of the secondary server to see whether or not it is running.

Within the secondary server's MariaDB shell, execute the following command:

```
SHOW SLAVE STATUS \G;
```

Here, we're adding \G, which changes the output to be displayed vertically instead of horizontally.

Assuming all went well, we should see the following line in the output:

```
Slave_IO_State: Waiting for master to send event
```

If the secondary server isn't running (Slave_IO_State is blank), execute the following command:

```
START SLAVE;
```

Next, check the status of the secondary server process again to verify:

```
SHOW SLAVE STATUS \G;
```

From this point forward, any data you add to your database on the primary server should be replicated to the secondary. To test, add a new record to the Employees table on the mysampledb database on the primary server:

```
USE mysampledb;
INSERT INTO Employees VALUES ('Optimus Prime', '100', 'Transformer');
```

On the secondary server, check the same database and table for the new value to appear. It may take a second or two:

```
USE mysampledb;
SELECT * FROM Employees;
```

If you see any errors in the Slave_IO_State line when you run `SHOW SLAVE STATUS \G`, or your databases aren't synchronizing properly, here are a few things you can try. First, make sure that the primary database server is listening for connections on `0.0.0.0` port 3306.

To test this, run this variation of the `ss` command to see which port the `mariadb` process is listening on (it's listed as `mysqld`):

```
ss -tulpn | grep mysqld
```

The output should be similar to the following:

```
tcp LISTEN 0      80          0.0.0.0:3306
0.0.0.0:*      users:(("mysqld",pid=9768,fd=23))
```

If you see that the service is listening on `127.0.0.1:3306` instead, that means it's only accepting connections from `localhost`. Earlier in this section, I mentioned changing the `bind` address in the `/etc/mysql/mariadb.conf.d/50-server.cnf` file. Make sure you've already done that and restart `mariadb`. During my tests, I've actually had one situation where the `mariadb` service became locked after I made this change, and attempting to restart the process did nothing (I ended up having to reboot the entire server, which is not typically something you'd have to do). Once the server came back up, it was listening for connections from the network.

If you receive errors on the secondary server when you run `SHOW SLAVE STATUS \G`, with regards to authentication, make sure you've run `FLUSH PRIVILEGES` on the primary server. Even if you have, run it again to be sure. Also, double-check that you're synchronizing with the correct username, IP address, and password. For your convenience, here's the command we ran on the primary server to grant replication permissions:

```
GRANT REPLICATION SLAVE ON *.* to 'replicate'@'192.168.1.204' identified by
'password';
FLUSH PRIVILEGES
```

Here's the command that we ran on the secondary server:

```
CHANGE MASTER TO MASTER_HOST="192.168.1.184", MASTER_USER='replicate', MASTER_
PASSWORD='password';
```

Finally, make sure that your primary database and the secondary database both contain the same databases and tables. The primary server won't be able to update a database on the secondary server if it doesn't exist there. Flip back to my example usage on `mysqldump` if you need a refresher. You should only need to use `mysqldump` and import the database onto the secondary server once, since after you get the replication going, any changes made to the database on the primary server should follow over to the secondary. If you have any difficulties with the `mysqldump` command, you can manually create `mysamp1edb` and the `Employees` table on the secondary server, which is really all it needs for synchronization to start.

Synchronization should then begin within a minute, but you can execute `STOP SLAVE`, followed by `START SLAVE`, on the secondary server to force it to try to synchronize again without waiting.

And that should be all there is to it. At this point, you should have fully functional primary and secondary database servers at your disposal. To get additional practice, try adding additional databases, tables, and users, and insert new rows into your databases.

It's worth mentioning that the users we've created here will not be synced to the secondary server, so you can use the commands we used earlier in this chapter to create users on the secondary server if you wish for them to be present there.

Summary

Depending on your skill set, you're either an administrator who is learning about SQL databases for the first time, or you're a seasoned veteran who is curious about how to implement a database server with Ubuntu Server. In this chapter, we dove into Ubuntu's implementation of this technology and worked through setting up our own database server. We also worked through some examples of the MariaDB syntax, such as creating databases, as well as setting up users and their grants. We also worked through setting up primary and secondary servers for replication.

Database administration is a vast topic, and we've only scratched the surface here. Being able to manage MySQL and MariaDB databases is a very sought-after skill for sure. If you haven't worked with these databases before, this chapter will serve as a good foundation for you to start your research.

In the next chapter, we'll use our database server to act as a foundation for Nextcloud, which we will set up as part of our look into setting up a web server. When you've finished practicing these database concepts, head on over to *Chapter 14, Serving Web Content*, where we'll journey into the world of web hosting.

Further reading

- Configuring MariaDB with Option Files:
<https://mariadb.com/kb/en/configuring-mariadb-with-option-files/>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



14

Serving Web Content

The flexible nature of Ubuntu Server makes it an amazing platform on which to host your organization's web presence. In this chapter, we'll take a look at Apache and NGINX, which make up the leading web server software on the internet. We'll go through installing, configuring, and extending both, as well as securing them with **Transport Layer Security (TLS)**. In addition, we'll also take a look at installing Nextcloud, which is a great solution for setting up your very own cloud environment for your organization to use for collaboration and sharing files. As we work through concepts related to hosting web content on Ubuntu Server, we will cover:

- Installing and configuring Apache
- Installing additional Apache modules
- Securing Apache with TLS
- Installing and configuring NGINX
- Setting up and configuring Nextcloud

To get us started, we'll first look at configuring Apache, as well as some basic configuration.

Installing and configuring Apache

The best way to become familiar with any technology is to dive right in. We'll begin this chapter by installing Apache. But first, what exactly is Apache? For those that aren't already aware, Apache is a popular application that is typically run on Linux and Unix servers to serve web pages to users. It runs in the background, and serves HTML pages to those that request a URL that exists on your server.

Installing Apache is very easy; it's simply a matter of installing the `apache2` package:

```
sudo apt install apache2
```

By default, Ubuntu will immediately start and enable the `apache2` daemon as soon as its package is installed. You can confirm this yourself with the following command:

```
systemctl status apache2
```


In fact, at this point, you already have (for all intents and purposes) a fully functional web server. If you were to open a web browser and enter the IP address of the server you just installed Apache on, you should see Apache's sample web page:

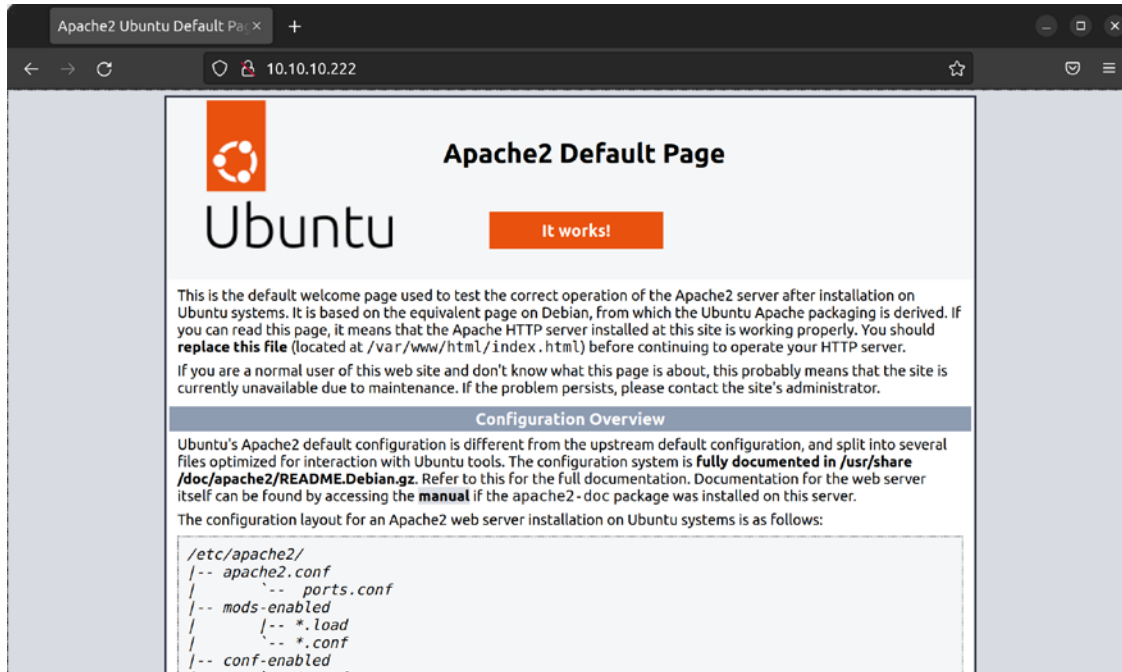


Figure 14.1: The default sample web page provided by Apache

There you go, you have officially served web content. All you needed to do was install the apache2 package, and your server was transformed into a web server. Chapter over, time to move on.

Of course, there's more to Apache than simply installing it and having it present a sample web page. While you could certainly replace the content in the sample web page with your own and be all set when it comes to hosting content for your users, there's much more to understand. For instance, there are several configuration files in the `/etc/apache2` directory that govern how sites are hosted, as well as which directories Apache will look in to find web pages to host. Apache also features plugins, which we will go over as well.

The directory that Apache serves web pages from is known as the **document root**, with `/var/www/html` being the default. Inside that directory, you'll see an `index.html` file, which is actually the default page you see when you visit an unmodified Apache server. Essentially, this is a test page that is designed to show you that the server is working, as well as some tidbits of information regarding the default configuration.

You're not limited to hosting just one website on a server, though. Apache supports the concept of a **virtual host**, which allows you to serve multiple websites from a single server. Each virtual host consists of an individual configuration file, which differentiates itself based on either name or IP address.

For example, you could have an Apache server with a single IP address that hosts two different websites, such as `acmeconsulting.com` and `acmesales.com`. These are hypothetical websites, but you get the idea. To set this up, you would create separate configuration files for `acmeconsulting.com` and `acmesales.com` and store them in your Apache configuration directory. Each configuration file would include a `<VirtualHost>` stanza, where you would place an identifier such as a name or IP address that differentiates one from the other. When a request comes in, Apache will serve either `acmeconsulting.com` or `acmesales.com` to the user's browser, depending on which criteria matched when the request came in. The configuration files for each site typically end with the `.conf` filename extension and are stored in the `/etc/apache2/sites-available` directory. We'll go over all of this in more detail shortly, so don't worry if you don't understand this yet.

The basic workflow for setting up a new site (virtual host) will typically be similar to the following:

- The web developer creates the website and related files
- These files are uploaded to Ubuntu Server, typically in a subdirectory of `/var/www` or another directory the administrator has chosen
- After adding the necessary files into the Document Root directory, the administrator will make sure that the `www-data` user owns all of the files within (in the case of Apache)
- The server administrator creates a configuration file for the site and copies it into the `/etc/apache2/sites-available` directory
- The administrator enables the site and reloads Apache

Enabling virtual hosts is handled a bit differently in Debian and Ubuntu than on other platforms. In fact, there are two specific commands to handle this purpose: `a2ensite` for enabling a site and `a2dissite` for disabling a site. You won't find these commands on distributions such as CentOS, for example. Configuration files for each site are stored in the `/etc/apache2/sites-available/` directory, and we would use the `a2ensite` command to enable each configuration. Assuming a site with the URL `acmeconsulting.com` is to be hosted on our Ubuntu server, we would create the `/etc/apache2/sites-available/acmeconsulting.com.conf` configuration file and enable the site with the following commands:

```
sudo a2ensite acmeconsulting.com.conf
sudo systemctl reload apache2
```

I'm not using absolute paths in my examples; as long as you've copied the configuration file to the correct place, the `a2ensite` and `a2dissite` commands will know where to find it.

If we wanted to disable the site for some reason, we would execute the `a2dissite` command against the site's configuration file:

```
sudo a2dissite acmeconsulting.com.conf
sudo systemctl reload apache2
```

If you're curious about how this works behind the scenes, when the `a2ensite` command is run against a configuration file, it basically creates a symbolic link to that file and stores it in the `/etc/apache2/sites-enabled` directory. When you run `a2dissite` to disable a site, this symbolic link is removed.

Apache, by default, will use any configuration files it finds in the `/etc/apache2/sites-enabled` directory. After enabling or disabling a site, you'll need to refresh Apache's configuration, which is where the `reload` option comes in. This command won't restart Apache itself (so users who are using your existing sites won't be disturbed) but it does give Apache a chance to reload its configuration files. If you replace `reload` with `restart` in the preceding commands, Apache will perform a full restart. You should only need to do that if you're having an issue with Apache or enabling a new plugin, but in most cases the `reload` option is preferred on a production system.

The main configuration file for Apache is located at `/etc/apache2/apache2.conf`. Feel free to view the contents of this file; the comments contain a good overview of how Apache's configuration is laid out. The following lines in this file are of special interest:

```
# Include the virtual host configurations:
IncludeOptional sites-enabled/*.conf
```

As you can see, this is how Ubuntu has configured Apache to look for enabled sites in the `/etc/apache2/sites-enabled` directory. Any file stored there with the `.conf` file extension is read by Apache. If you wish, you could actually remove those lines and Apache would then behave as it does on other platforms, and the `a2ensite` and `a2dissite` commands would no longer have any purpose. However, it's best to keep the framework of Ubuntu's implementation intact, as separating the configuration files makes logical sense and helps simplify the configuration. This chapter will go along with the Ubuntu way of managing configuration.

An additional virtual host is not required if you're only hosting a single site. The contents of `/var/www/html` are served by the default virtual host if you make no changes to Apache's configuration. This is where the example site that ships with Apache comes from. If you only need to host one site, you could remove the default `index.html` file stored in this directory and replace it with the files required by your website. If you wish to test this for yourself, you can make a backup copy of the default `index.html` file and create a new one with some standard HTML. You should see the default page change to feature the content you just added to the file.

The `000-default.conf` file is special, in that it's basically the configuration file that controls the default Apache sample website. If you look at the contents of the `/etc/apache2/sites-available` and `/etc/apache2/sites-enabled` directories, you'll see the `000-default.conf` configuration file stored in `sites-available` and symlinked in `sites-enabled`. This shows you that, by default, this site was included with Apache, and its configuration file was enabled as soon as Apache was installed. For all intents and purposes, the `000-default.conf` configuration file is all you need if you only plan on hosting a single website on your server. The contents of this file are as follows, but I've stripped the comments out of the file in order to save space on this page:

```
<VirtualHost *:80>
    ServerAdmin webmaster@localhost
    DocumentRoot /var/www/html
```

```
ErrorLog ${APACHE_LOG_DIR}/error.log
CustomLog ${APACHE_LOG_DIR}/access.log combined
</VirtualHost>
```

As you can see, this default virtual host is telling Apache to listen on port 80 for requests and to serve content from `/var/www/html` as soon as requests come in. The `<VirtualHost>` declaration at the beginning is listening to everything (the asterisk is a *wildcard*) on port 80, so this is basically handling all web traffic that comes into the server from port 80. The `ServerAdmin` clause specifies the email address that is displayed in any error messages shown if there is a problem with the site.

The `DocumentRoot` setting tells Apache which directory to look for in order to find files to serve as connections to this virtual host. `/var/www/html` is the default, but some administrators choose to customize this. This file also contains lines for where to send logging information. The `access log` contains information relating to HTTP requests that come in, which by default is stored in `/var/log/access.log`. The `error log` is stored at `/var/log/error.log` and contains information you can use whenever someone has trouble visiting your site. The `${APACHE_LOG_DIR}` variable equates to `/var/log` by default, and this is set in the `/etc/apache2/envvars` file, in case for some reason you wish to change this (for example, you wish to use a custom logging directory).

If you wish to host another site on the same server by creating an additional virtual host, you can use the same framework as the original file, with some additional customizations. Virtual host files are stored in the `/etc/apache2/sites-available` directory, with a filename ending in `.conf`. Here's an example of a hypothetical website, `acmeconsulting.com`. A virtual host file such as this might be saved as `/etc/apache2/sites-available/acmeconsulting.com.conf`:

```
<VirtualHost 192.168.1.104:80>
  ServerAdmin webmaster@localhost
  DocumentRoot /var/www/acmeconsulting

  ErrorLog ${APACHE_LOG_DIR}/acmeconsulting.com-error.log
  CustomLog ${APACHE_LOG_DIR}/acmeconsulting.com-access.log combined
</VirtualHost>
```

To save time, I'll generally copy another virtual host file, even the default one, and change it accordingly. In this particular example, I've emphasized some important differences. First, with this virtual host, I'm not listening for all connections coming in on port 80; instead, I'm specifically looking for incoming traffic going to IP address `192.168.1.104` on port 80. This works because this server has two network cards, and therefore two IP addresses. With virtual hosts, I'm able to serve a different website, depending on which IP address the request is coming in on.

Next, I set `DocumentRoot` to `/var/www/acmeconsulting`. Each virtual host should have its own individual `DocumentRoot` to keep each site separate from the others. On my servers, I will typically disable or remove the sample virtual host (the one that has the default `DocumentRoot` of `/var/www/html`). Instead, I use `/var/www` as a base directory, and each virtual host gets its own directory as a subdirectory of this base.

Another change I find useful is to give each virtual host its own log files. Normally, Apache will use `/var/log/apache2/error.log` and `/var/log/apache2/access.log` to store log entries for all sites. If you only have a single site on your server, that is fine. However, when you're serving multiple sites, I find it useful to give each site its own independent log files. That way, if you are having trouble with a particular site, you don't have to scroll through unrelated log entries to find what you're looking for when you're troubleshooting. In my example, I inserted the website name in the log filenames, so this virtual host is logging errors in the `/var/log/apache2/acmeconsulting.com-error.log` file, and the access log is being written to `/var/log/apache2/acmeconsulting.com-access.log`. These log files will be created for you automatically as soon as you reload Apache.

With a server that only has a single IP address, you can still set up multiple virtual hosts. Instead of differentiating virtual hosts by IP, you can instead differentiate them by name. This is common on **Virtual Private Server (VPS)** installations of Ubuntu, where you'll typically have a single IP address assigned to you by your VPS provider. For name-based virtual hosts, we would use the `ServerName` option in our configuration. Refer to the following example to see how this would work. With this example, I'm adding name-based virtual hosts to their own files. I called mine `000-virtual-hosts.conf` and stored it in the directory. The contents are as follows:

```
<VirtualHost *:80>
    ServerName acmeconsulting.com
    DocumentRoot /var/www/acmeconsulting
</VirtualHost>

<VirtualHost *:80>
    ServerName acmesales.com
    DocumentRoot /var/www/acmesales
</VirtualHost>
```

For each virtual host, I'm declaring a `ServerName` with a matching `DocumentRoot`. With the first example, any traffic coming into the server requesting `acmeconsulting.com` will be provided a `DocumentRoot` of `/var/www/acmeconsulting`. The second example looks for traffic from `acmesales.com` and directs it to `/var/www/acmesales`. You can list as many virtual hosts here as you'd like to host on your server. Providing your server has enough resources to handle traffic to each site, you can host as many as you need.

If you're using domain names with virtual hosts, then this will only work if you set up networking such that the domain name referenced in the file resolves to the IP address of your server. Depending on your configuration, there are multiple ways to do this. If you're using a VPS provider, such as DigitalOcean or Linode, your server will have an IP address already and you only need to edit the **A Record** on your DNS server to point to that IP. (The various types of DNS entries, such as an A Record, were covered in *Chapter 11, Setting Up Network Services*.)

If you're running your own DNS server, you would add the A Record there. If you're using an external DNS provider, you would log in to the dashboard for your account and add the A Record there. For testing purposes, you can edit your `/etc/hosts` file on your local workstation (not the server) to point to your new web server. If you're not using a VPS provider, you would need to forward port 80 in your firewall to point to your internal web server. This is beyond the scope of this book, as there are many different models of firewalls available and it's impossible to cover them all.

As we continue through this chapter, we'll perform some additional configurations for Apache. At this point though, you should have an understanding of the basics of how Apache is configured in Ubuntu Server. For extra practice, feel free to create additional virtual hosts and serve different pages for them.

Installing additional Apache modules

Apache features additional modules that can be installed that will extend its functionality. These modules can provide additional features such as adding support for things like Python or PHP. Ubuntu's implementation of Apache includes two specific commands for enabling and disabling modules, `a2enmod` and `a2dismod`, respectively. Apache modules are generally installed via packages from Ubuntu's repositories. To see a list of modules available for Apache, run the following command:

```
apt search libapache2-mod
```

In the results, you'll see various module packages available, such as `libapache2-mod-python` (which adds Python support) and `libapache2-mod-php8.1` (which adds PHP 8.1 support), among many others. Installing an Apache module is done the same way as any other package, with the `apt install` command. In the case of PHP support, we can install the required package with the following command:

```
sudo apt install libapache2-mod-php8.1
```

Installing a module package alone is not enough for a module to be usable in Apache, though. Modules must be enabled in order for Apache to be able to utilize them. As mentioned earlier, we can use the `a2enmod` and `a2dismod` commands for respectively enabling or disabling a module. You can view a list of modules that are built into Apache with the following command:

```
apache2 -l
```

The modules shown in the output will be those that are built into Apache, so you won't need to enable them. If the module your website requires is listed in the output, you're all set.

To view a list of all modules that are installed and ready to be enabled, you can run the `a2enmod` command by itself with no options:

```

jay@ubuntu-server: ~
Try: sudo apt install <deb name>
jay@ubuntu-server:~$ a2enmod
Your choices are: access_compat actions alias allowmethods asis auth_basic auth_digest
auth_form authn_anon authn_core authn_dbd authn_dbm authn_file authn_socache
authnz_fcgi authnz_ldap authz_core authz_dbd authz_dbm authz_groupfile authz_host
authz_owner authz_user autoindex brotli buffer cache cache_disk cache_socache cern
_meta cgi cgid charset lite data dav dav_fs dav_lock dbd deflate dialup dir dump
echo env expires ext_filter file_cache filter headers heartbeat heartmonitor http
p2 ident imagemap include info lbmethod_bybusyness lbmethod_byrequests lbmethod_by
traffic lbmethod_heartbeat ldap log_debug log_forensic lua macro md mime mime_magi
c mpm_event mpm_prefork mpm_worker negotiation proxy proxy_ajp proxy_balancer prox
y_connect proxy_express proxy_fcgi proxy_fdpass proxy_ftp proxy_hcheck proxy_html
proxy_http proxy_http2 proxy_scgi proxy_uwsgi proxy_wstunnel ratelimit reflector r
emoteip reqtimeout request rewrite sed session session_cookie session_crypto sessi
on_dbd setenvif slotmem_plain slotmem_shm socache_dbm socache_memcache socache_red
is socache_shmcb speling ssl status substitute suexec unique_id userdir usertrack
vhost_alias xml2enc
Which module(s) do you want to enable (wildcards ok)?

```

Figure 14.2: The `a2enmod` command showing a list of available Apache modules

The end of the output of the `a2enmod` command will ask you whether or not you'd like to enable any of the modules:

```
Which module(s) do you want to enable (wildcards ok)?
```

If you wanted to, you could type the names of any additional modules you'd like to enable and then press `Enter`. Alternatively, you can press `Enter` without typing anything to simply return to the prompt.

If you give the `a2enmod` command a module name as an option, it will enable it for you. To enable PHP 8.1 (which we'll need later), you can run the following command:

```
sudo a2enmod php8.1
```

Chances are, though, if you've installed a package for an additional module, it was most likely enabled for you during installation. With Debian and Ubuntu, it's very common for daemons and modules to be enabled as soon as their packages are installed, and Apache is no exception. In the case of the `libapache2-mod-php8.1` package I used as an example, the module should've been enabled for you once the package was installed:

```
sudo a2enmod php8.1
```

If a module is already enabled, you will see output similar to the following when you try to enable it with `a2enmod`:

```
Module php8.1 already enabled
```


If the module wasn't already enabled, we would see the following output:

```
Enabling module php8.1.  
To activate the new configuration, you need to run:  
systemctl restart apache2
```

As instructed, we'll need to restart Apache in order for the enabling of a module to take effect. Keep in mind that restarting Apache will make any sites it may host become unavailable during the process. When it comes to disabling a module, the command syntax is fairly similar. To do so, you'll use the `a2dismod` command along with the name of the module you'd like to disable:

```
sudo a2dismod php8.1
```

Enabling a module that was already previously enabled will result in output similar to the following:

```
Module php8.1 disabled.
```

To activate the new configuration, you need to run:

```
systemctl restart apache2
```

The modules you install and enable on your Apache server will depend on the needs of your website. For example, if you're going to need support for Python, you'll want to install the `libapache2-mod-python` package. If you're installing a third-party package, such as WordPress or Drupal, you'll want to refer to the documentation for those packages in order to obtain a list of which modules are required for the solution to install and run properly. Once you have such a list, you'll know which packages you'll need to install and which modules to enable.

Securing Apache with TLS

Nowadays, it's a great idea to ensure your organization's website is encrypted and available over HTTPS. Encryption of web traffic has been historically achieved by utilizing **Secure Sockets Layer (SSL)** or, more recently, **TLS**, which is the successor to SSL. Both refer to a method of utilizing cryptography by installing signed certificates that protect and encrypt web traffic. The two functions are different, but the end result is the same. Going forward, it's recommended to use TLS due to the additional security strength it offers, though it's not uncommon to see SSL being used nowadays since it hasn't been completely phased out.

Setting up and benefiting from TLS is not all that difficult to do, and will help protect your organization against common vulnerabilities being potentially exploited. Utilizing TLS doesn't protect you from all exploits being used in the wild, but it does offer a layer of protection you'll want to benefit from. Not only that, but your customers pretty much expect you to secure their communications nowadays. In this section, we'll look at how to use TLS with our Apache installation. We'll work through enabling it, generating certificates, and configuring Apache to use those certificates with both a single-site configuration and with virtual hosts.

By default, Ubuntu's Apache configuration listens for traffic on port 80, but not port 443 (HTTPS). You can check this yourself by running the following command:

```
sudo ss -tulpn | grep apache
```

The results will look similar to the following and will show the ports that Apache is listening on, which is only port 80 by default:

```
tcp    LISTEN 0      511          *:80
*:*    u
sers:(("apache2",pid=11521,fd=4),("apache2",pid=11520,fd=4),
("apache2",pid=11519,fd=4),("apache2",pid=11518,fd=4),("apache2",pid=11517,fd=4),
("apache2",pid=11513,fd=4))
```

If the server were listening on port 443 as well, we would've seen the following within the output:

```
tcp    LISTEN 0      511          *:443      *:*
```

To enable support for HTTPS traffic, we need to first enable the `ssl` module:

```
sudo a2enmod ssl
```

Next, we need to restart Apache:

```
sudo systemctl restart apache2
```

In addition to the sample website we discussed earlier, Ubuntu's default Apache implementation also includes another site configuration file, `/etc/apache2/sites-available/default-ssl.conf`. Unlike the sample site, this one is not enabled by default. This configuration file is similar to the sample site configuration, but it's listening for connections on port 443 and contains additional configuration items related to TLS. Here's the content of that file, with the comments stripped out in order to save space on this page:

```
<IfModule mod_ssl.c>
    <VirtualHost _default_:443>
        ServerAdmin webmaster@localhost

        DocumentRoot /var/www/html

        ErrorLog ${APACHE_LOG_DIR}/error.log
        CustomLog ${APACHE_LOG_DIR}/access.log combined

        SSLEngine on

        SSLCertificateFile      /etc/ssl/certs/ssl-cert-
snakeoil.pem
        SSLCertificateKeyFile /etc/ssl/private/ssl-cert-
```

```
        snakeoil.key

        <FilesMatch ".(cgi|shtml|phtml|php)$">
            SSLOptions +StdEnvVars
        </FilesMatch>
        <Directory /usr/lib/cgi-bin>
            SSLOptions +StdEnvVars
        </Directory>

    </VirtualHost>
</IfModule>
```

We already went over the `ServerAdmin`, `DocumentRoot`, `ErrorLog`, and `CustomLog` options earlier in this chapter, but there are additional options in this file that we haven't seen yet. On the first line, we can see that this virtual host is listening on port 443. We also see `_default_` listed here instead of an IP address. The `_default_` option only applies to unspecified traffic, which in this case means any traffic coming into port 443 that hasn't been identified in any other virtual host. In addition, the `SSL` engine option enables TLS traffic. Right after that, we have options for our TLS certificate file and key file, which we'll get to a bit later.

We also have a `<Directory>` clause, which allows us to apply specific options to a directory. In this case, the `/usr/lib/cgi-bin` directory is having the `SSLOptions +StdEnvVars` settings applied, which enables default environment variables for use with TLS. This option is also applied to files that have an extension of `.cgi`, `.shtml`, `.phtml`, or `.php` through the `<FilesMatch>` option. The `BrowserMatch` option allows you to set options for specific browsers, though it's out of scope for this chapter. For now, just keep in mind that if you want to apply settings to specific browsers, you can.

By default, the `default-ssl.conf` file is not enabled. In order to benefit from its configuration options, we'll need to enable it, which we can do with the `a2ensite` command as we would with any other virtual host:

```
sudo a2ensite default-ssl.conf
```

Even though we just enabled TLS, our site isn't secure just yet. We'll need TLS certificates installed in order to secure our web server. We can do this in one of two ways, with self-signed certificates or certificates signed by a certificate authority. Both are implemented in very similar ways, and I'll discuss both methods. For the purposes of testing, self-signed certificates are fine. In production, self-signed certificates would technically work, but most browsers won't trust them by default and will give you an error when you go to their page. Therefore, it's a good idea to refrain from using self-signed certificates on a production system. Users of a site with self-signed certificates would need to bypass an error page before continuing to the site, and seeing this error may cause them to avoid your site altogether. You can install the certificates into each user's web browser, but that can be a headache. In production, it's best to use certificates signed by a vendor.

Another method of setting up a certificate on your server is **Let's Encrypt**, a popular (and free) service for encrypting web traffic. Consider checking out the instructions at the Let's Encrypt website at letsencrypt.org/docs, as well as the example article mentioned at the end of the chapter.

As we go through this process, I'll first walk you through setting up TLS with a self-signed certificate so you can see how the process works. We'll create the certificate and then install it into Apache. You won't necessarily need to create a website to go through this process, since you could just secure the sample website that comes with Apache if you wanted something to use as a proof of concept. After we complete the process, we'll take a look at installing certificates that were signed by a certificate authority.

To get the ball rolling, we'll need a directory to house our certificates. I'll use `/etc/apache2/certs` in my examples, although you can use whatever directory you'd like, as long as you remember to update Apache's configuration with your desired location and filenames:

```
sudo mkdir /etc/apache2/certs
```

For a self-signed certificate and key, we can generate the pair with the following command. Feel free to change the name of the key and certificate files to match the name of your website:

```
sudo openssl req -x509 -nodes -days 365 -newkey rsa:2048 -keyout /etc/apache2/certs/mysite.key -out /etc/apache2/certs/mysite.crt
```

You'll be prompted to enter some information for generating the certificate. Answer each prompt as they come along. Here's a list of the questions you'll be asked, along with my responses for each. Change the answers to fit your server, environment, organization name, and location:

```
Country Name (2 letter code) [AU]:US
State or Province Name (full name) [Some-State]:Michigan
Locality Name (eg, city) []:Detroit
Organization Name (eg, company) [Internet Widgits Pty Ltd]:My Company
Organizational Unit Name (eg, section) []:IT
Common Name (e.g. server FQDN or YOUR name) []:myserver.mydomain.com
Email Address []:webmaster@mycompany.com
```

Now, you should see that two files have been created in the `/etc/apache2/certs` directory, `mysite.crt` and `mysite.key`, which represent the certificate and private key, respectively. Now that these files have been generated, the next thing for us to do is to configure Apache to use them. Look for the following two lines in the `/etc/apache2/sites-available/default-ssl.conf` file:

```
SSLCertificateFile      /etc/ssl/certs/ssl-cert-snakeoil.pem
SSLCertificateKeyFile  /etc/ssl/private/ssl-cert-snakeoil.key
```

Comment these lines out by placing a `#` symbol in front of both:

```
# SSLCertificateFile      /etc/ssl/certs/ssl-cert-snakeoil.pem
# SSLCertificateKeyFile  /etc/ssl/private/ssl-cert-snakeoil.key
```

Next, add the following two lines underneath the lines you just commented out. Be sure to replace the target directories and certificate filenames with yours, if you followed your own naming convention:

```
SSLCertificateFile      /etc/apache2/certs/mysite.crt
SSLCertificateKeyFile  /etc/apache2/certs/mysite.key
```

To make Apache benefit from the new configuration, reload the apache2 daemon:

```
sudo systemctl reload apache2
```

With the new configuration in place, we're not quite done but we're close. We still have a small bit of configuration left to add. But before we get to that, let's return to the topic of installing TLS certificates that were signed by a certificate authority. The process for installing signed TLS certificates is pretty much the same, but the main difference is how the certificate files are requested and obtained. Once you have them, you will copy them to your file server and configure Apache the same way as we just did. To start the process of obtaining a signed TLS certificate, you'll need to create a **Certificate Signing Request (CSR)**. A CSR is basically a request for a certificate in file form that you'll supply to your certificate authority to start the process of requesting a signed certificate. A CSR can be easily generated with the following command:

```
openssl req -new -newkey rsa:2048 -nodes -keyout server.key -out server.csr
```

With the CSR file that was generated, you can request a signed certificate. The CSR file should now be in your current working directory. The entire process differs from one provider to another, but in most cases, it's fairly straightforward. You'll send them the CSR, pay their fee, fill out a form or two on their website, prove that you are the owner of the website in question, and then the vendor will send you the files you need. It may sound complicated, but certificate authorities usually walk you through the entire process and make it clear what they need from you in order to proceed. Once you complete the process, the certificate authority will send you your certificate files, which you'll then install on your server. Once you configure the `SSLCertificateFile` and `SSLCertificateKeyFile` options in `/etc/apache2/sites-available/default-ssl.conf` to point to the new certificate files and reload Apache, you should be good to go.

There's one more additional step we should perform for setting this up properly. At this point, our certificate files should be properly installed, but we'll need to inform Apache of when to apply them. If you recall, the `default-ssl.conf` file provided by the `apache2` package is answering requests for any traffic not otherwise identified by a virtual host (the `<VirtualHost _default_:443>` option). We will need to ensure that our web server is handling traffic for our existing websites when TLS is requested. We can add a `ServerName` option to that file to ensure our site supports TLS.

Add the following option to the `/etc/apache2/sites-available/default-ssl.conf` file, right underneath `<VirtualHost _default_:443>`:

```
ServerName mydomain.com:443
```

Now, when traffic comes into your server on port 443 requesting a domain that matches the domain you typed for the `ServerName` option, it should result in a secure browsing session for the client. You should see the green padlock icon in the address bar (this depends on your browser), which indicates that your session is secured. If you're using self-signed certificates, you'll probably see an error you'll have to skip through first, and you may not get the green padlock icon. This doesn't mean the encryption isn't working; it just means your browser is skeptical of the certificate since it wasn't signed by a known certificate authority. Your session will still be encrypted.

If you are planning on hosting multiple websites over HTTPS, you may want to consider using a separate virtual host file for each. An easy way to accomplish this is to use the `/etc/apache2/sites-available/default-ssl.conf` file as a template and change `DocumentRoot` to the directory that hosts the files for that site. In addition, be sure to update the `SSLCertificateFile` and `SSLCertificateKeyFile` options to point to the certificate files for the site and set `ServerName` to the domain that corresponds to your site. Here's an example virtual host file for a hypothetical site that uses TLS. I've highlighted lines that I've changed from the normal `default-ssl.conf` file:

```
<IfModule mod_ssl.c>
    <VirtualHost *:443>
        ServerName acmeconsulting.com:443

        ServerAdmin webmaster@localhost

        DocumentRoot /var/www/acmeconsulting
        ErrorLog ${APACHE_LOG_DIR}/acmeconsulting.com-error.log
        CustomLog ${APACHE_LOG_DIR}/acmeconsulting.com-access.
        log combined

        SSLEngine on

        SSLCertificateFile /etc/apache2/certs/acmeconsulting/acme.
        crt
        SSLCertificateKeyFile /etc/apache2/certs/acmeconsulting/acme.
        key

        <FilesMatch ".(cgi|shtml|phtml|php)$">
            SSLOptions +StdEnvVars
        </FilesMatch>
        <Directory /usr/lib/cgi-bin>
            SSLOptions +StdEnvVars
        </Directory>

    </VirtualHost>
</IfModule>
```

Basically, what I did was create a new virtual host configuration file (using the existing `default-ssl.conf` file as a template). I called this new file `acme-consulting.conf` and I stored it in the `/etc/apache2/sites-available` directory. I changed the `VirtualHost` line to listen for anything coming in on port 443. The line `ServerName acmeconsulting.com:443` was added to make this file responsible for traffic coming in looking for `acmeconsulting.com` on port 443. I also set `DocumentRoot` to `/var/www/acmeconsulting`. In addition, I customized the error and access logs so that it will be easier to find log messages relating to this new site, since its log entries will go to their own specific files.

In my experience, I find that a modular approach, such as what I've done with the sample virtual host file for HTTPS, works best when setting up a web server that's intended to host multiple websites. With each site, I'll typically give it its own document root, certificate files, and log files. Even if you're only planning on hosting a single site on your server, using this modular approach is still a good idea, since you may want to host additional sites later on.

So, there you have it. You should now understand how to set up secure virtual hosts in Apache.

Installing and configuring NGINX

Apache isn't the only technology that is capable of allowing you to host web content on your server. NGINX also serves the same purpose and is gaining popularity quite rapidly. Apache is still a great option, and even if that's your chosen web server software, it's a good idea to at least be familiar with NGINX and learn its basics. NGINX itself is a proxy server as well but is capable of also serving web content, which is why it competes with Apache.

Before we do so, I want to mention first that you can really only have one web server service running on a single web server. If you've been following along up to now, you currently have a functional Apache web server. If you were to also install NGINX, it probably wouldn't start as the ports it wants to listen on (port 80 and/or 443) will already be in use. You can run both on a single server, but that's outside the scope of this book. Ideally, you'd want to use one or the other. Therefore, to continue with this section you'd either want to remove Apache or set up a separate web server for testing NGINX. I recommend the latter, because later on in this chapter we will take a look at hosting Nextcloud, and we will be using Apache to do so. If you remove Apache now, you'd have to add it back in order to follow along with that section. Theoretically, you'd only have to stop the `apache2` process before starting `nginx`, but the two resources sharing the same server have a lot of variables and may conflict.

To get started with NGINX, simply install it:

```
sudo apt install nginx
```

Just like with Apache, if we enter the IP address of our server in a browser, we're presented with a sample page, but this time NGINX's version instead of the one that ships with Apache. It certainly looks boring in comparison, but it works:



Figure 14.3: The NGINX sample page

The default configuration files for nginx are stored in the `/etc/nginx` directory. Go ahead and peruse these files to get a general feel for how the configuration is presented. Similar to Apache, you also have `sites-enabled` and `sites-available` directories here, which serve the same purpose.

Just as with Apache, the `sites-available` directory houses configuration files for sites that *can be* enabled, while the `sites-enabled` directory stores configuration files for sites that are enabled. Unlike Apache, though, we don't have dedicated commands to enable these sites. We have to link them manually. Although we haven't even looked at NGINX configuration files yet, let's just assume that we have created the following configuration file:

```
/etc/nginx/sites-available/acmesales.com
```

To enable that site, we would need to create a symbolic link for it and store that link in the `/etc/nginx/sites-enabled` directory:

```
sudo ln -s /etc/nginx/sites-available/acmesales.com /etc/nginx/sites-enabled/
acmesales.com
```

Then, we can reload nginx:

```
sudo systemctl reload nginx
```

As it stands right now, a site configuration file named `default` exists in `/etc/nginx/sites-available` and a symbolic link to it is already present in `/etc/nginx/sites-enabled`. If all we want to do is host a single site, we only need to replace the default content that NGINX serves, which is located in the `/var/www/html` directory (the same as Apache) with the content for our site. After refreshing the page, we're good to go.

If we want to serve more than one site from one server, the default file is a great starting point for creating additional virtual hosts. We can start by copying it to a new name:

```
sudo cp /etc/nginx/sites-available/default /etc/nginx/sites-available/  
acmesales.com
```

Obviously, `acmesales.com` is an example, so feel free to name this whatever you wish.

Now, we can edit this file and change it to serve additional content. First of all, only one site can be referred to as a *default* site. A default site in NGINX is one that answers if none of the other sites match a request. Therefore, we want to remove both occurrences of `default_server` from our newly copied config. Find these lines:

```
listen 80 default_server;  
listen [::]:80 default_server;
```

Change them to this:

```
listen 80;  
listen [::]:80;
```

Next, we'll need to adjust the `server_name` option to refer to the name of our new site. Add this line:

```
server_name acmesales.com www.acmesales.com;
```

Now, we'll need to change the document root to the directory that will store the files for our new site. Find this line:

```
root /var/www/html;
```

And change it to this:

```
root /var/www/acmesales.com;
```

The final file should look like the following at this point:

```
server {  
    listen 80;  
    listen [::]:80;  
  
    root /var/www/acmesales.com;  
  
    index index.html index.htm index.nginx-debian.html;  
  
    server_name acmesales.com www.acmesales.com;  
  
    location / {  
        try_files $uri $uri/ =404;  
    }  
}
```


You can probably see that the configuration format for NGINX configuration files is simpler than with Apache. I find this to be true, and I've noticed that sites I've configured with NGINX generally have fewer lines in their configuration files than Apache does.

At this point, assuming that you have the required content in `/var/www/acmesales.com` and have a proper configuration file, the new site should respond as soon as you reload `nginx`. But what about TLS? I recommend that we always secure our websites, regardless of which solution we're using to serve it. With NGINX, we can add that feature easily. The certificate files themselves are the same regardless of whether we're using Apache or NGINX. If you haven't already created your certificate files, refer back to the section in this chapter where we did so. Assuming you already have certificate files, we just need to make additional changes to our configuration.

First, we change the first two lines to listen on port 443 with TLS instead of standard port 80:

```
listen 443 ssl;
listen [::]:443 ssl;
```

Next, we'll add the following two lines before the `location` section:

```
ssl_certificate /etc/certs/cert.pem;
ssl_certificate_key /etc/certs/cert.key;
ssl_session_timeout 5m;
```

For this to work, you'll need to adjust the paths and the names of the cert files to make sure they match what you called them on your server. The entire file should look similar to the following at this point:

```
server {
    listen 443 ssl;
    listen [::]:443 ssl;

    root /var/www/html;

    index index.html index.htm index.nginx-debian.html;

    server_name acmesales.com www.acmesales.com;

    ssl_certificate /etc/certs/cert.pem;
    ssl_certificate_key /etc/certs/cert.key;
    ssl_session_timeout 5m;
    location / {
        try_files $uri $uri/ =404;
    }
}
```

Finally, a potential problem is that users may access our site via port 80, instead of utilizing HTTPS. We can tell NGINX to forward these people to the secure version of our site automatically. To do that, we can edit the default configuration file (`/etc/nginx/sites-available/default`) and add the following line just after the two `listen` directives:

```
return 301 https://$host$request_uri;
```

Now, anytime a user visits the HTTP version of our site, they'll be redirected to the secure HTTPS version automatically.

Now that we've looked at serving web content with both Apache and NGINX, let's take a look at setting up our very own Nextcloud server.

Setting up and configuring Nextcloud

I figured we'd end this chapter with a fun activity: setting up our very own Nextcloud server. Nextcloud is a very useful web application that's handy for any organization. Even if you're not working on a company network, Nextcloud is a great asset for even a single user. You can use it to synchronize files between machines, store and sync contacts, keep track of tasks you're working on, fetch email from a mail server, and more. To complete this activity, you'll need a web server to work with. Nextcloud supports multiple different web server platforms, but in this example, we'll be using Apache.

You'll also need an installation of MySQL or MariaDB, as Nextcloud will need its own database. We went over installing and managing MariaDB databases in *Chapter 13, Managing Databases*. I'll give you all the commands you'll need to set up the database in this section, but refer back to *Chapter 13, Managing Databases*, if any of these commands confuse you.

To get started, we need to download Nextcloud. To do so, head on over to the project's website at <https://www.nextcloud.com> and navigate to the **Download** section. The layout of this site may change from time to time, but at the time of writing, the first link to click on is a button labeled **Get Nextcloud**, which will take you to the download page. As of the time of writing, the direct URL for the download page for Nextcloud is currently <https://nextcloud.com/install>.

Once there, look for the **Archive File** heading, and expand it. Underneath that, you should see a button that reads **Download for Server**, but don't click on it just yet. Instead, right-click on it and click **Copy link address** or a similarly named option, depending on the browser you use.

This should copy the link for the download to your clipboard. If the Nextcloud website layout has changed since publication, you're essentially just looking for the URL to the ZIP file to download Nextcloud, and you'll want to copy it to your computer's clipboard.

Next, open an SSH session to your web server. Make sure you're currently working from your home directory, and execute the following command:

```
wget <URL of Nextcloud>
```

To get the Nextcloud URL, simply paste the URL into your terminal after typing `wget`. Your entire command will look similar to the following:

```
wget https://download.nextcloud.com/server/releases/latest.zip
```

This command will download the Nextcloud software locally to your current working directory. Next, we'll need to unzip the archive:

```
unzip latest.zip
```

If you get an error message insinuating that the `unzip` command is not available, you may need to install it:

```
sudo apt install unzip
```

Now, let's move the newly extracted `nextcloud` directory to `/var/www/html`:

```
sudo mv nextcloud /var/www/html/nextcloud
```

In order for Nextcloud to function, the user account that Apache uses to serve content will need full access to it. Let's use the following command to give the user `www-data` ownership of the `nextcloud` directory:

```
sudo chown www-data:www-data -R /var/www/html/nextcloud
```

Now, you should have the required files for the Nextcloud software installed on the server in the `/var/www/nextcloud` directory. In order for this to work, though, Apache will need a configuration file that includes `/var/www/nextcloud` as its document root. We can create the file we need at the following location:

```
/etc/apache2/sites-available/nextcloud.conf
```

Example content to include in that file is as follows:

```
Alias /nextcloud "/var/www/html/nextcloud/"

<Directory /var/www/html/nextcloud/>
    Options +FollowSymlinks
    AllowOverride All

    <IfModule mod_dav.c>
        Dav off
    </IfModule>

    SetEnv HOME /var/www/html/nextcloud
    SetEnv HTTP_HOME /var/www/html/nextcloud
</Directory>
```

Similar to our earlier discussion on Apache, we're adding a config file here specifically for Nextcloud that sets up an alias to point to /nextcloud to `www.mydomain.com/nextcloud`. Essentially, it allows Nextcloud to be reached at your domain name, plus /nextcloud added to the end. The rest of the file disables WebDAV (a means of allowing a web server to act as a file server, not needed in our case) and then enables environment variables to set HOME and HTTP_HOME to the document root for Nextcloud, which in our case is `/var/www/html/nextcloud`.

Next, we enable the new site:

```
sudo a2ensite nextcloud.conf
```

Next, we'll need to make a change to Apache. First, we'll need to ensure that the `libapache2-mod-php8.1` package is installed since Nextcloud requires PHP, but we'll need some additional packages as well. You can install Nextcloud's prerequisite packages with the following command:

```
sudo apt install libapache2-mod-php8.1 php8.1-curl php8.1-gd php8.1-intl  
php8.1-mbstring php8.1-mysql php8.1-xml php8.1-zip
```

Next, restart Apache so that it can take advantage of the new PHP plugin:

```
sudo systemctl restart apache2
```

At this point, we'll need a MySQL or MariaDB database for Nextcloud to use. This database can exist on another server, or you can share it on the same server you installed Nextcloud on. If you haven't already set up MariaDB, a walk-through was covered during *Chapter 13, Managing Databases*. At this point, it's assumed that you already have MariaDB installed and running.

Log in to your MariaDB instance as your root user, or a user with full root privileges. You can create the Nextcloud database with the following command:

```
CREATE DATABASE nextcloud;
```

Next, we'll need to add a new user to MariaDB for Nextcloud and give that user full access to the nextcloud database. We can take care of both with the following command:

```
GRANT ALL ON nextcloud.* to 'nextcloud'@'localhost' IDENTIFIED BY 'super_  
secret_password';
```

Make sure to change `super_secret_password` to a very strong (preferably randomly generated) password. Make sure you save this password in a safe place.

Now we have all we need in order to configure Nextcloud. You should now be able to visit your Nextcloud instance in a web browser. Just enter a URL similar to the following, replacing the sample IP address with the one for your server:

```
http://172.16.250.133/nextcloud
```

If you're using a subdomain and gave Nextcloud its own virtual host, that URL would then be something like this:

```
http://nextcloud.yourdomain.com/nextcloud
```

You should see a page asking you to configure Nextcloud:

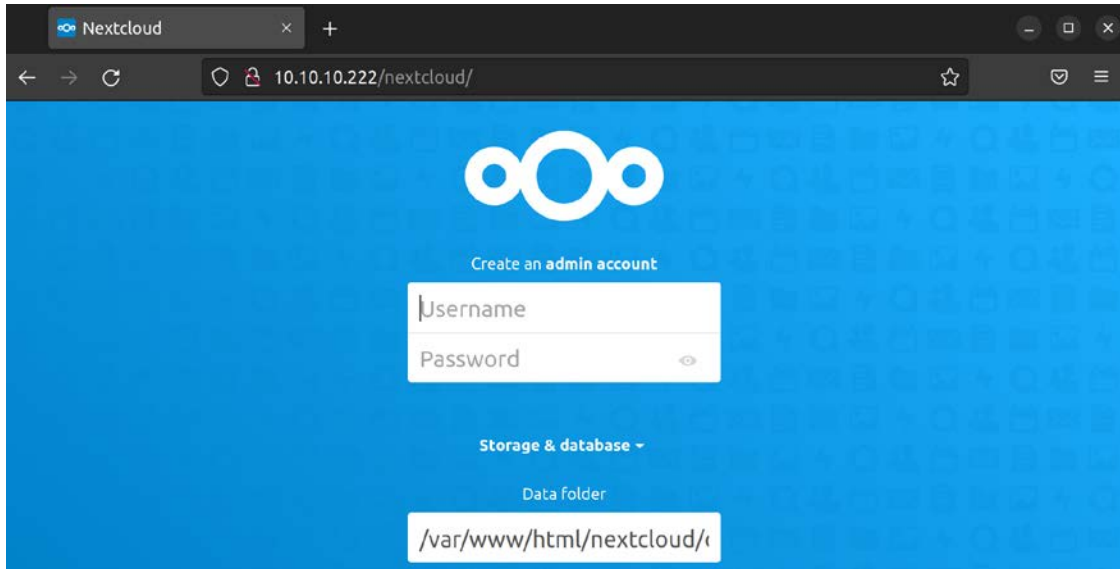


Figure 14.4: Nextcloud configuration page

If you do not see this page, make sure that the `/var/www/html/nextcloud` directory is accessible via Apache. Also, make sure you have an appropriate virtual host for Nextcloud referencing this directory as its Document Root.

This page will ask you for several pieces of information. First, you'll see **Username** and **Password**. This is not asking you for a pre-existing account, but actually to set up a brand-new administrator account. This shouldn't be an account you'll use on a day-to-day basis, but instead an admin account you'll use only when you want to add users and maintain your system. Please note that it won't ask you to confirm the password, so you'll want to make certain you're entering the password you think you are. It's perhaps safer to type the password in a text editor, and then copy and paste the password into the **Password** box to make sure you don't lock yourself out.

Data folder will default to `/var/www/html/nextcloud/data`. This default is normally fine, but if you have configured your server to have a separate data partition, you can configure that here. If you plan on storing a large amount of data on your Nextcloud server, setting up a separate partition for it may be a good idea. If you do, you can set that here. Otherwise, leave the default.

In the next section, you'll be asked to fill in information for the Nextcloud database we created earlier. **Database user** and **Database password** will use the values we created when we set up the MariaDB database for Nextcloud earlier. In my examples, I used `nextcloud` for the username as well as **Database name**. The password will be whatever it is you used for the password when we set up the database user account and granted privileges. Finally, the database server defaults to `localhost`, which is correct as long as you set up the database on the same machine as the Nextcloud server. If not, enter the address of your database server here, if it's somewhere else. The following screenshot shows the initial form completely filled out with the example values we've used so far in this section:

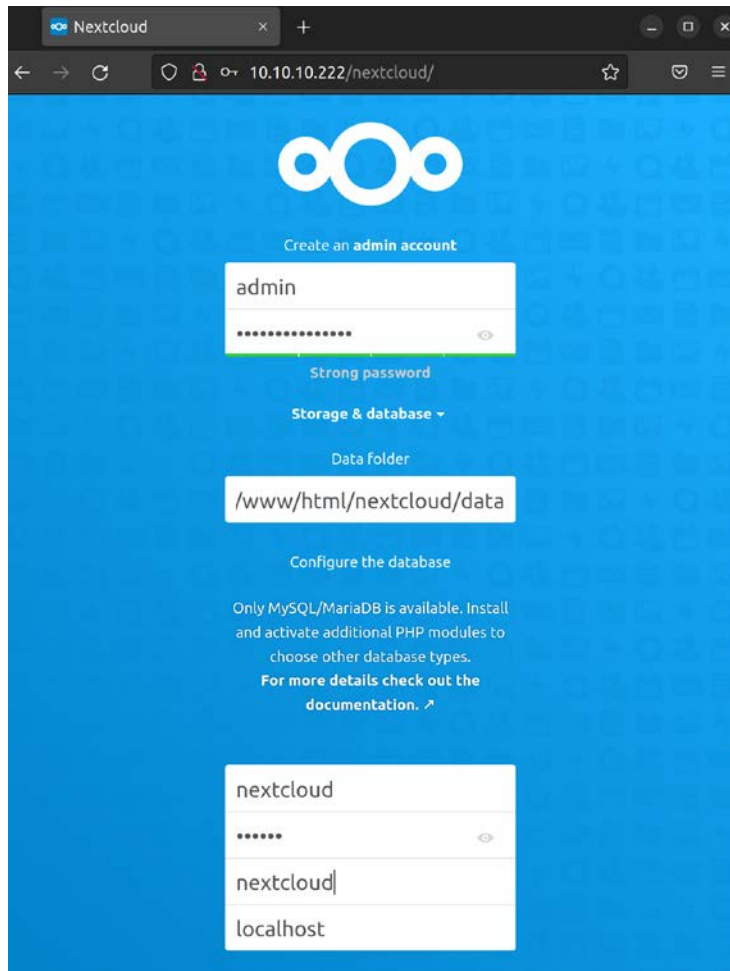


Figure 14.5: Nextcloud configuration page

That's it! Assuming all went well, Nextcloud will set itself up in the background and you'll then continue to the main screen. Since you only created an admin account so far, I recommend you create an account for yourself, as well as any friends or colleagues you'd like to check out your Nextcloud server. To do so, go to the top-right corner of the Nextcloud page, where it shows an icon that resembles a gear. When you click on this icon, you'll see an option for **Users**:

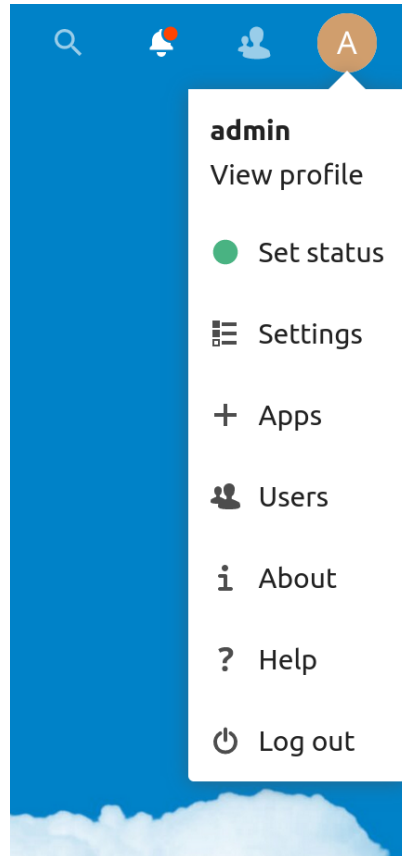
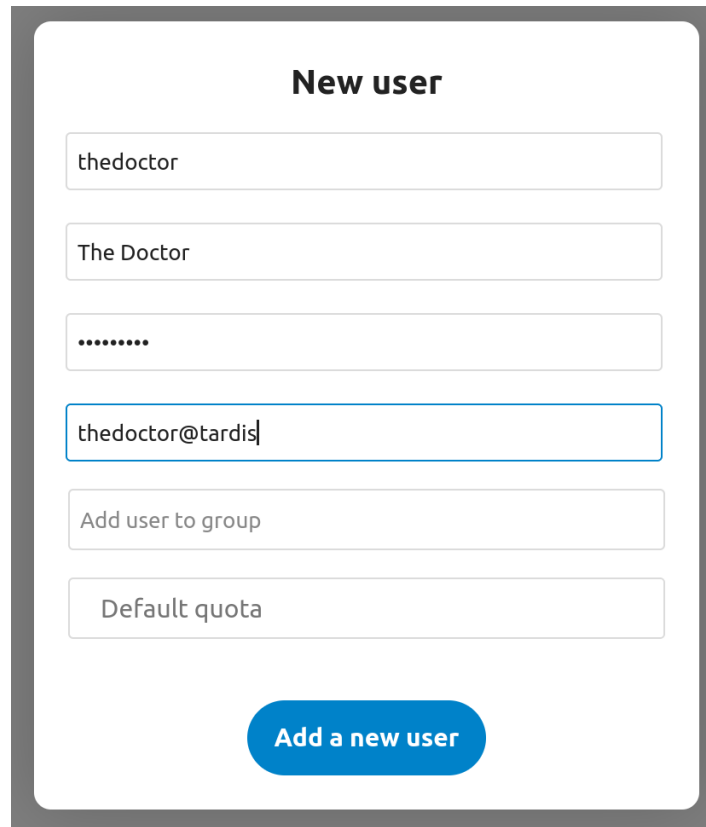


Figure 14.6: Nextcloud menu

On the **Users** screen, you'll be able to add additional users to access Nextcloud. Click on the **New User** button. Simply fill out the **Username** and **Password** fields at the top of the screen, and then click on the blue checkmark icon to finish the process:



The screenshot shows a 'New user' form with the following fields and values:

- Username: thedoctor
- Full name: The Doctor
- Password: (masked)
- Email: thedoctor@tardis
- Add user to group: (empty)
- Default quota: (empty)

A blue button labeled 'Add a new user' is located at the bottom of the form.

Figure 14.7: Adding a new user to Nextcloud

As an administrative user, you can enable or disable various apps that are used by your users. Out of the box, Nextcloud has a basic suite of apps enabled, such as the **File Sharing** and **Photos** plugins. There are many more apps that you can enable in order to extend its functionality. In the top-right corner of the main screen of Nextcloud, you'll find an icon that looks like a gear, and if you click on it, you will find a link to **Apps**, which will allow you to add additional functionality. Feel free to enable additional apps to extend Nextcloud's capabilities. Some of my must-haves include **Notes** and **Tasks**.

Now, you have your very own Nextcloud server. I find Nextcloud to be a very useful platform. Some Linux desktop environments (such as GNOME) have built-in integrations that allow you to add your Nextcloud account right to your desktop, which will allow calendar and contact syncing with your computer. There are also standalone client apps available for Linux, Windows, and macOS that you can download from the same URL that we've downloaded Nextcloud itself from.

I'm sure you'll agree that Nextcloud is a very useful asset to have available. For more information on using Nextcloud, check out the manual. In fact, it's available in the **Files** app from within the application itself.

Summary

In this action-packed chapter, we looked at serving web pages with Apache. We started out by installing and configuring Apache, and then added additional modules. We also covered the concept of virtual hosts, which allow us to serve multiple websites on a single server, even if we only have a single network interface. Then, we walked through securing our Apache server with TLS. With Apache, we can use self-signed certificates, or we can purchase TLS certificates from a vendor for a fee. We looked at both possibilities. We even set up NGINX, which is a very powerful application that is growing in popularity. Finally, we closed out the chapter with a guide to installing Nextcloud, which is an application I'm sure you'll find incredibly useful.

In the next chapter of our journey, we'll take a look at the process of automating server configuration with Ansible, which is a lot of fun.

Relevant videos

- Nextcloud setup walk-through: <https://linux.video/nextcloud-setup>

Further reading

- NGINX documentation: <https://nginx.org/en/docs/>
- NGINX documentation from the Ubuntu community wiki: <https://help.ubuntu.com/community/Nginx>
- Apache HTTP server documentation: <https://httpd.apache.org/docs/>
- Nextcloud administration manual: https://docs.nextcloud.com/server/latest/admin_manual/
- Certbot instructions: <https://certbot.eff.org/instructions>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



15

Automating Server Configuration with Ansible

Nowadays, it's not uncommon to have hundreds of servers that make up your organization's infrastructure. As our user base grows, we're able to scale our environment to meet the demands of our customers. As we scale our resources and add additional servers, the amount of time we spend configuring them and setting them up increases considerably. The time spent setting up new servers can be a major burden—especially if we need to create hundreds of servers within a small window of time. As workload demands increase, we need to have a solution in place to manage our infrastructure and quickly deploy new resources with as small a workload as possible. In this chapter, we explore the concept of configuration management along with automated deployments. This sure sounds complicated, but it's not—you'll be surprised how easy it is to automate your configuration.

In this chapter, we will cover:

- Understanding the need for configuration management
- Why Ansible?
- Creating a Git repository
- Getting started with Ansible
- Making your servers do your bidding
- Putting it all together – automating web server deployment
- Using Ansible's pull method

In the introduction, I've already given you some examples of why you might want to consider building automation into your workflow and implement an effective solution in your environment. In the next section, we'll explore the need for automation in more detail before we actually get started.

Understanding the need for configuration management

When I first started working in the IT industry, it was a much different landscape than it is today. Servers were all physical, and any time you needed a new server, you literally needed to call a vendor and order one.

You waited for a week or two for the server to be built and sent to you. When it arrived, you installed it in a rack, set up an operating system, and then installed whatever applications you needed. You then tested the server for a while to make sure the combination of software, hardware, and drivers was stable and reliable. After some time, you'd deploy the new server into production.

Nowadays, it's still the case that system administrators often need to purchase and install hardware, much like the process I mentioned in the previous paragraph. However, with virtual machines and containers, the physical hardware we install is commonly just a catalyst to host virtual resources. In the past, we had one physical server for each use case, which meant we needed to have very large server rooms. But in modern times, you may have a server with dozens of cores that are capable of running hundreds of virtual machines. But the problem of configuration remains—the process of setting up an operating system and applications is a very time-consuming endeavor.

As the landscape changed, the need for automation increased. Servers needed to be deployed quickly and efficiently. With the large number of servers in a typical data center, it became less and less practical to connect to each of them and configure them one by one every time a change was necessary. For example, when a security vulnerability hit the news, the typical administrator would need to manually install a patch on every server. This could take days or even weeks. That's not very efficient.

To better deal with this issue, the concept of configuration management has become very popular. With configuration management, an administrator can write some sort of code (such as a script) and then use a utility to execute it across every server. Configuration management is also known as **Infrastructure as Code (IaC)** and basically lets the administrator define a set of guidelines for servers of various types and have them automatically be provisioned to meet those requirements. This automation saves a ton of work.

Configuration management also comes into play while provisioning a new server. Imagine defining some rules for a specific type of server, and having it come to life meeting those exact specifications. The applications you want it to have are installed during the provisioning process, configuration files are copied over, users are created, and firewall rules are put in place, all automatically as defined in your specification. Put even more simply, imagine setting up something like a web server with just a single command. No need to install Apache or do any of that manual work. You simply request a server, and the configuration management solution you have in place will take care of the rest.

IaC, which is basically a fancy term for configuration management, is essentially just the automated running of scripts on your servers. In this book, we've looked at automation already. In *Chapter 6, Boosting Your Command-line Efficiency*, we wrote a simple script that we could use to back up a server. That same mentality can be used for provisioning servers as well, by simply having a server run a script when it comes online. For existing servers, you can make a change once and apply that change to every server you manage, or even just a subset.

This is where configuration management utilities, such as Chef, Puppet, and others, come into play. Each of these solutions features a specific type of scripting language that is designed from the ground up to facilitate the provisioning of resources. With such utilities, there is typically some sort of program (or locally installed agent) that interprets the instructions from a central server and runs them on its clients. Each solution is relatively smart; it will determine what needs to be done and perform the steps.

If a requirement is met, the instruction is skipped. If a required resource is not present, it will be configured appropriately. One such configuration management solution is Ansible, which we will use in this chapter.

Why Ansible?

In this chapter, I will show you how to set up Ansible, and then we will use it to automate some configuration tasks. By the end of this chapter, you'll understand the basic concepts you can use to start the process of automating deployments in your organization. You may be wondering, then, why Ansible and not one of the other solutions, such as Chef or Puppet?

Some configuration management solutions are relatively heavy from a resource perspective. With other solutions, you'll generally have a central server, which will run a master program. This program will periodically *check in* with each server under its control by communicating with the agent installed on each server. Then, the agent will receive instructions from the central server and carry them out.

This means that you'll need to maintain a server with modest CPU and RAM requirements, and the agent on the client side of the communication will also spend valuable CPU in order to carry out the instructions. This resource utilization can be very heavy on both the primary and client servers.

Ansible is very different than the other solutions in that there is no agent at all. There is typically a server, but it's not required to run any resource-intensive software. The entire configuration process happens via SSH, so you can even carry out the instructions from your workstation if you want to skip having to maintain a central server. Typically, the administrator will create a user account on each server, and then the central Ansible server (or workstation) will execute commands over SSH to update the configuration on each machine. Since there is no agent installed on each server, the process takes a lot less CPU. Of course, the instructions that Ansible gives your servers will definitely result in CPU usage, but certainly a lot less than the other solutions.

Ansible is typically set up by creating an **inventory file**, which contains a list of resources (servers) (in the form of hostnames or IP addresses that Ansible will be instructed to connect to and configure). If you want to add a new server, you simply make sure that a specific user account exists on that server, then you add it to the inventory list. If you want to remove it, you delete the line in the inventory file corresponding to that server. It's very easy.

However, something that's magical about Ansible is that you don't even have to run a central server at all if you don't want to. You can store your Ansible configuration in a Git repository, and have each server download code from the repository and run it locally. This means that if you do have a dynamic environment where servers come and go all the time (which is very common in cloud deployments), you don't have to worry about maintaining an inventory file. Just instruct each server to download the code and provision themselves. This is known as the **pull method** of Ansible, which I will also show you.

While solutions such as Chef and Puppet have their merits and are definitely fun to use, I think you'll find that Ansible scales better and gives you far more control over how these hosts are configured. While it's up to you to figure out exactly how you want to implement Ansible, the creative freedom it gives you is second to none. I've been using Ansible for quite a while now, and I'm still finding new ways to use it. It's a technology that will grow with you.

Creating a Git repository

For the examples in this chapter, it's recommended that you create a Git repository to store your Ansible code. This isn't required, as you can find other ways of hosting your code, but it's highly recommended. This is especially true when we get to the pull method of Ansible at the end of this chapter. In this section, I'll walk you through creating a repository. If you already know how to use GitHub, you can skip this section.

While a full walkthrough of Git is beyond the scope of this book, the basics are more than enough for following along here. When it comes to Git, you can simply install the `git` package on a server to have it host your code, but GitHub is probably the easiest way to get started. An added bonus is that GitHub is home to a lot of great projects you can benefit from, and browsing the code of these projects is a great way to become more accustomed to syntax rules with different scripting and programming languages. For our purposes, we're going to use GitHub as a central place to store our Ansible code.



It probably goes without saying, but GitHub is a public resource. Any code you upload to the service will be available for all to see by default. Therefore, be mindful of the information you commit to your repository. Make sure you don't include any personally identifiable information, passwords, blueprints, API keys, or anything else you don't want the public to know about you or your organization. You can create a private repository to hide confidential information, but it's still safer to not upload protected information at all (whether the repository is private or not).

To get started, create an account at <https://www.github.com> if you don't already have one. This is a free process. Make sure you create a reasonably secure password here. After you have created an account, click on **New repository**, and then give it a name (simply calling it `ansible` is fine):

Create a new repository

A repository contains all project files, including the revision history. Already have a project repository elsewhere? [Import a repository](#).


Owner * **Repository name ***


 LearnLinuxTV ▾ / ansible 

Great repository names are short and memorable. Need inspiration? How about [shiny-octo-train?](#)

Description (optional)

My awesome repository for ansible configs

 **Public**
Anyone on the internet can see this repository. You choose who can commit.

 **Private**
You choose who can see and commit to this repository.

Intialize this repository with:
Skip this step if you're importing an existing repository.


Add a README file
This is where you can write a long description for your project. [Learn more.](#)

Add .gitignore
Choose which files not to track from a list of templates. [Learn more.](#)

.gitignore template: None ▾

Choose a license
A license tells others what they can and can't do with your code. [Learn more.](#)

License: None ▾

 You are creating a public repository in your personal account.

[Create repository](#)

Figure 15.1: Creating an Ansible repository on GitHub

In the example screenshot, I created a repository that is **Public**, which means exactly that—anyone will be able to view the code. You can create **Private** repositories as well, if you prefer. Since we're not going to include confidential information in the repository during the examples in this book, we don't need to worry about that right now.

Once the repository is created using **Create repository**, we'll need to download it locally. For that, we'll need to install the `git` package:

```
sudo apt install git
```

Next, we should set up our local Git client so that we can fill out our name and email address, otherwise Git will most likely complain. To do this, we can issue the following commands, substituting the content in the quotes with your information:

```
git config user.email "you@example.com"  
git config user.name "John Doe"
```

To download our repository, the following will do the trick (ignore the warning about the repository being empty if you see such a message):

```
git clone https://github.com/myusername/ansible.git
```

Now you have the Git repository downloaded locally. Right now, the repository doesn't include anything useful. To create a file within the repository, you simply change your working directory to be inside the repository folder that was downloaded when you cloned it, and create whatever files you want inside. By default, Git doesn't care about any files you create inside the repository until you add them. For example, we can create a test file and commit it to the repository with the following commands:

```
echo "this is a test" > testfile.txt  
git add testfile.txt  
git commit -m "initial commit"
```

With these commands, we used `echo` to create a test file with some text in it. Then, we used the `git add` command to tell Git that we want the file to be a part of our repository. Finally, we finalized our changes by using `git commit`, along with the `-m` flag and a message about the commit. At this point, the changes only exist locally. To push our changes back to GitHub, we use the following command from inside the repository directory:

```
git push origin main
```

By following the on-screen prompts (GitHub username and password), our changes will be placed inside our actual repository.

So, how does this help us when it comes to configuration management? Typically, the code that administrators use to provision servers will be placed inside a repository for safekeeping. If the local copy of the code is lost, it can simply be cloned again from the repository. GitHub is a safe place to put our code, since we can be reasonably sure that our code won't disappear as the service is very stable (you still may want to create a local backup to be safe). Whether you're using Ansible, Chef, Puppet, or another utility, it's common practice to keep the code in a Git repository. With regard to Ansible, this will directly impact us in the last section of this chapter since we'll be using the `ansible-pull` command, which expects a repository URL.

In practice, when we create Ansible playbooks, you should commit those changes back to the repository. I won't specifically call on you to do that, so go ahead and keep that in mind as we go. When you create a new playbook, add it to the repository, then commit it. If you make changes to existing files, commit those changes. Be sure to use the `git push` command to push your changes back to the repository. For example, if you created a file inside the repository named `myplaybook.yml`, you would execute commands such as these:

```
git add myplaybook.yml
git commit -m "insert message about the commit here"
git push origin main
```

Go ahead and practice this a bit before you move on. Even if you don't use Ansible in production, understanding the basics of Git is invaluable, as you'll almost definitely need it at some point in your career.

Getting started with Ansible

The first thing to know about Ansible is that it changes constantly. New versions with exciting features are released regularly, and it shows no sign of slowing down whatsoever. There is a lot of excitement around this technology, so it's regularly improving.

The reason I'm bringing this up is that although the examples in this book have been written for (and tested on) Ubuntu 22.04, new versions of Ansible are released regularly outside of Ubuntu's repositories, and not only include new features but syntax changes as well. For our needs in this book, the version of Ansible included in the repositories should be more than fine. However, if you were to look at examples of Ansible playbooks online, they might be written for a newer version (or even an older version). If for any reason you have an issue with a particular example written for Ansible, a good first step in terms of troubleshooting is to compare the version of Ansible the tutorial was written for against the version installed. When it comes to Ubuntu 22.04, the version of Ansible that's made available is 2.10.x. You can install a newer version of Ansible from the official website, but for the purposes of this book, we'll use the version that's in the default repositories.

Go ahead and install `ansible` via `apt`:

```
sudo apt install ansible
```

Now you should have the `ansible-playbook` command available, which is the main binary that we'll be using as we explore Ansible. There are other commands that Ansible provides us, but we're not concerned with them.

In order to follow along with the remainder of this chapter, it's recommended that you have at least two servers to work with; the more the better. If you have a **Virtual Machine (VM)** solution such as VirtualBox available, simply create additional VMs. To save time, consider cloning an existing VM a few times (just make sure you don't overload your computer/server by over-allocating resources).

The most common workflow of Ansible works something like this: you have a main server or workstation, on which Ansible is installed. While you don't need an agent on the clients, they will, however, need OpenSSH installed and configured as that's how Ansible communicates. To make things easy, it's recommended to have a dedicated Ansible user on each machine, and the Ansible user on the server should be able to connect to each machine without a password. It doesn't matter what you call the Ansible user; you can simply use `ansible` or something clever. We covered how to create SSH keys in *Chapter 10, Connecting to Networks*, so refer to that if you need a reminder. Creating users was covered in *Chapter 2, Managing Users and Permissions*. In a nutshell, here are the things you should work on in order to set up your environment for Ansible:

1. Install Ansible on a central server or workstation
2. Create a user for Ansible on each machine you want to manage configuration on
3. Create the same user on your central server or your local machine
4. Set up the Ansible user on the server so that it can connect to clients via SSH without a password
5. Configure `sudo` on the client machines so that the Ansible user can execute commands with `sudo` with no password

In previous chapters, we covered how to create users and SSH keys, but we have yet to cover the last point. Assuming you named your Ansible user `ansible`, create the following file:

```
/etc/sudoers.d/ansible
```

Inside that file, place the following text:

```
ansible ALL=(ALL) NOPASSWD: ALL
```

Next, we need to ensure that the file is owned by root:

```
sudo chown root:root /etc/sudoers.d/ansible
```

Finally, we need to adjust the permissions of the file:

```
sudo chmod 440 /etc/sudoers.d/ansible
```

Go ahead and test this out. On the server, switch to the `ansible` user:

```
sudo su - ansible
```

Then, to test this out, use SSH to execute a command on a remote machine:

```
ssh 192.168.1.123 sudo ls /etc
```

How does this work? You may or may not know this, but if you use SSH to execute just one command, you don't necessarily need to set up a persistent connection. In this example, we first switch to the `ansible` user. Then, we connect to `192.168.1.123` (or whatever the IP address of the client is) and tell it to execute `sudo ls /etc`. Executing an `ls` command with `sudo` may seem like a silly thing to do, but it's great—it allows you to test whether or not `sudo` works without doing anything potentially dangerous. Listing the contents of a directory is about as innocent as you can get.

It may seem like an awful lot of steps in order to get configuration management working. But make sure you think with a system administrator's mindset—these setup tips can be automated. In my case, I have a Bash script that I run on each of my servers that sets up the required user, keys, and sudo access specifically for Ansible. Anytime I want to add a new server to Ansible, I simply run that script on the machine just once, and from that point forward Ansible will take care of the rest.

What should have happened is that the command should have executed and printed the contents of `/etc` without prompting you for a password. If this doesn't work, make sure you have completed each of the recommended steps. You should have an `ansible` user on each machine, and that user should have access to `sudo` without a password since we created a file for that user in `/etc/sudoers.d`. If the SSH portion fails, check the log file at `/var/log/auth.log` for clues, as that is where related errors will be saved. Once you have met these requirements, we can get automating with Ansible!

Making your servers do your bidding

As server administrators, we're control freaks. There are few things more exciting than executing a command and having every single server obey it and carry it out. Now that we have Ansible set up, that's exactly what we're going to do. I'm assuming by now you have some machines you want to configure, and they're all set up to communicate via SSH with your central server. Also, as I mentioned before, I highly recommend you utilize something like Git to store your configuration files, but that's not required for this section.

Setting up an inventory file and configuring Ansible settings

First, we'll need an inventory file, which is a special text file Ansible expects to find that tells it where to find servers to connect to. In previous versions, the process of installing the `ansible` package would provide you with some default configuration, located in `/etc/ansible`. In Ubuntu 22.04, at least with the version of Ansible that's offered in the default repositories, this is no longer the case. For our purposes, that's fine, though – we were going to create an empty configuration anyway.

First, let's create the directory that we'll be using for the configuration on our “controller” server (the one we'll be using to control, or “configure,” other nodes):

```
sudo mkdir /etc/ansible
```

After that, we can create an empty inventory file:

```
sudo touch /etc/ansible/hosts
```

There's actually a way to avoid needing to create an inventory file, which we'll get into later in this chapter.

We should also make sure that only the Ansible user account can read it. Execute the following command to change ownership (replace `ansible` with whatever user account you chose as your Ansible account if you're using something different):

```
sudo chown ansible /etc/ansible/hosts
```

Next, modify the permissions such that only the owner can view or change the file:

```
sudo chmod 600 /etc/ansible/hosts
```

Next, let's edit the `/etc/ansible/hosts` file and populate it. If you did end up having a default hosts file created for you when you installed Ansible, you can simply empty the file since we'll be creating our own anyway. What we'll do with this file is populate it with the IP addresses of the servers we wish to manage. Feel free to make a backup of the original host file, if it already exists. For right now, we're only including the IP addresses of the nodes we want to control with Ansible, so the file will end up looking similar to the following:

```
192.168.1.145
192.168.1.125
192.168.1.166
```

That's it; it's simply just a list of IP addresses. I bet you were expecting some long configuration with all kinds of syntax requirements? Sorry to disappoint. All you need to do is copy a list of IP addresses of the servers you want to manage into this file. If you have DNS names set up for the machines you want to configure, you can use them instead:

```
myhost1.mydomain.com
myhost2.mydomain.com
myhost3.mydomain.com
```

Since Ansible understands IP addresses as well as DNS names, we can use either or a combination of both in order to set up our inventory file. We can also split up our hosts within the inventory file between different roles, but that is outside the scope of this book. I do recommend learning about roles in Ansible if you wish to take your knowledge further (see the *Relevant videos* section for more information).

If you decide not to store your inventory file at `/etc/ansible/hosts`, you must tell Ansible where to find it. There is another important file to Ansible, and that is its configuration file, located at `/etc/ansible/ansible.cfg`. Inside this file, we can fine-tune Ansible to get the best performance possible. While we won't go over this file in detail, just know that you can seriously increase the performance of Ansible by fine-tuning its settings, and Ansible will read settings from its configuration file every time it runs. In our case, if we wish to store our inventory file somewhere other than `/etc/ansible/hosts`, we will need to add the following two lines to this file (most likely, you'll need to create this file since it probably doesn't already exist at this point):

```
[defaults]
inventory = /path/to/hosts
remote_user = jay
```

With the first setting, we're basically telling Ansible where to find its inventory file. There are many more configuration items we can place in the `ansible.cfg` file to configure it further, but that's all we need to configure right now. With the second line, we're setting a default user to use for Ansible playbooks, which must exist on any servers you wish to manage with it.

Similar to the inventory file, Ansible also checks the local directory for a file named `ansible.cfg` to fetch its configuration, so you could actually include the configuration file in the Git repository as well, and then execute Ansible commands from within the repository directory. This works because Ansible will check for the existence of a configuration file in your current working directory, and use it if it's found there. You may want to be careful about including your configuration file in your Git directory, though. While it's not as private as the inventory file, it can potentially contain privileged information. Therefore, you may want to keep the file at `/etc/ansible/ansible.cfg` and manage it outside of the Git repository if you include anything private in the file (for example, encryption keys).

Now we can test out whether or not Ansible is working at this point. Thankfully, this is also easy. Just simply execute the following command:

```
ansible all -m ping
```

The results should look similar to this:

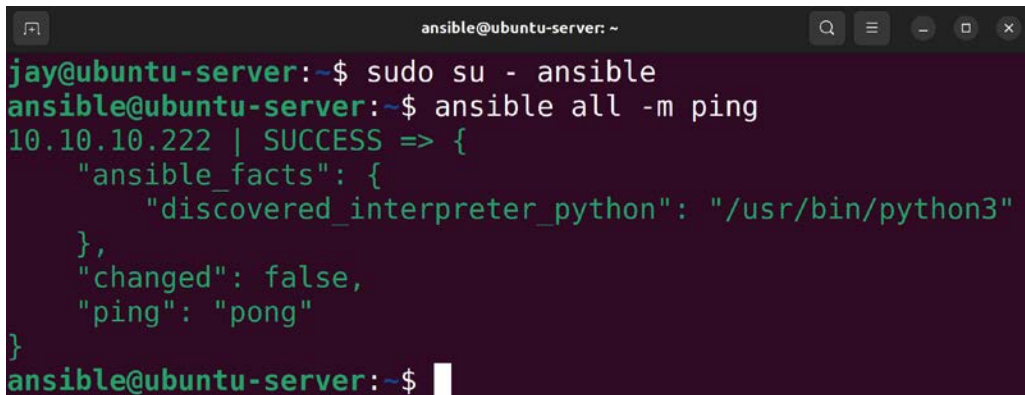
A terminal window titled 'ansible@ubuntu-server: ~' showing a user named 'jay' at 'ubuntu-server' running 'sudo su - ansible'. The prompt changes to 'ansible@ubuntu-server:~\$'. The user then runs 'ansible all -m ping'. The output shows a successful connection to 10.10.10.222, displaying a JSON object with 'ansible_facts' (including 'discovered_interpreter_python' as '/usr/bin/python3'), 'changed' as false, and 'ping' as 'pong'. The prompt returns to 'ansible@ubuntu-server:~\$'.

Figure 15.2: Testing Ansible

Depending on how many servers you have set up, you should see that output one or more times. You may think that all that command has done is a simple ping test, but `ping` means something different to Ansible than usual. It's actually attempting to make a connection to the server, to test its availability. If it fails, double-check that the hosts are available over SSH. Success here simply means that Ansible is able to communicate with your hosts via SSH. Now that the communication exists, we can build some actual configuration.

Configuring client servers

Ansible uses something called a **playbook** in order to store configuration. A playbook is essentially a file in the YAML format with instructions that Ansible understands, that are interpreted as a set of tasks run against hosts. A full guide on YAML isn't something we'll be covering here – you don't have to master this format or even fully understand it to use it with Ansible. That will automatically come in time. The takeaway here is that YAML is simply the format that Ansible uses, and it's a file format that's not specific to Ansible itself. A playbook is basically just a collection of instructions written in YAML format, and each individual instruction is known as a play.

You can think of this with the analogy of a sport, like football. Although I don't know the first thing about football, I do know that football coaches have playbooks containing things that they want their players to do, and each action by a player is a play. It's the same concept here.

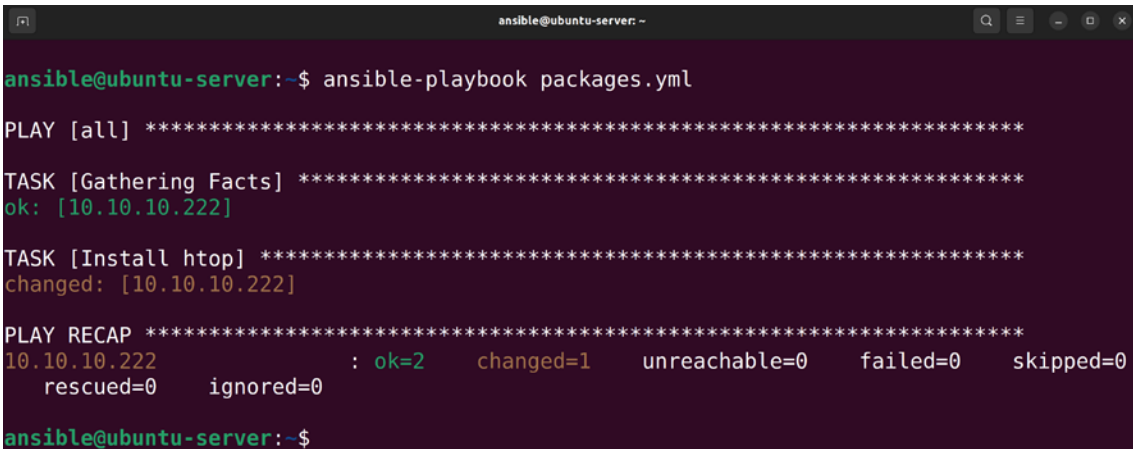
Let's write our first playbook. Create a file called `packages.yml` in your local Ansible directory. You can fill it with this content (make sure you include the hyphens):

```
---
- hosts: all
  become: true
  tasks:
    - name: Install htop
      ansible.builtin.apt:
        name: htop
```

We can run this playbook with the following command:

```
ansible-playbook packages.yml
```

This will produce an output that looks something like the following:



```
ansible@ubuntu-server:~$ ansible-playbook packages.yml
PLAY [all] *****
TASK [Gathering Facts] *****
ok: [10.10.10.222]
TASK [Install htop] *****
changed: [10.10.10.222]
PLAY RECAP *****
10.10.10.222 : ok=2  changed=1  unreachable=0  failed=0  skipped=0
             rescued=0  ignored=0
ansible@ubuntu-server:~$
```

Figure 15.3: Ansible in action

Just like that, all of the hosts in your inventory file will have the `htop` package installed. It really doesn't matter which package you install, so long as it exists in the repositories; I just used `htop` as a simple example. But once you run it, you should see an overview of what was changed. Ansible will tell you how many items your hosts updated, how many tasks have failed, and how many targets weren't reachable at the time the playbook was run.

Let's take a closer look at what the instructions in this sample playbook do. The hyphens at the beginning are part of the YAML format, so we really don't need to get into that. Spacing is very important with the YAML format, as you need to be consistent throughout.

In my examples, I am inserting two spaces underneath each heading. A heading starts with a hyphen:

```
- hosts: all
```

Here, we declare which hosts we want to have the commands apply to. I added `all` here, which basically runs the configuration against every host in the inventory file. With advanced usage, you can actually create roles in Ansible and divide your hosts between them, such as a web server, database server, and so on. Then, you can apply configuration only to hosts inside a particular role. We're not going to get into that in this chapter, but just know that it is possible.

The next line is become:

```
become: true
```

This line is basically Ansible's term for describing `sudo`. We're telling Ansible to use `sudo` to execute the commands, since installing packages requires root privileges. The next part of the playbook is as follows:

```
tasks:
```

This line starts the next section, which is where we place our individual tasks. Next, we name our new task (Ansible refers to a "task" as a "play"):

```
- name: Install htop
```

With `name`, we give the play a name. This isn't required but you should always include it. The importance of this is that whatever we type here is what is going to show up in the logs if we enable logging, and will also print to the terminal as the play runs. We should be descriptive here, as it will certainly help if a play fails and we need to locate it in a log file that has hundreds of lines. Next, we utilize the `apt` module and tell it to install a package, `htop` in this case:

```
ansible.builtin.apt:  
  name: htop
```

We use the `ansible.builtin.apt` module simply because Ubuntu uses the `apt` command to manage packages, but modules exist for all of the popular Linux distributions. Ansible's support of package managers among various distributions is actually fairly extensive. All of the major distributions, such as Red Hat, Fedora, openSUSE, Arch Linux, and Debian, are supported (and those are just the ones I've used in my lab off the top of my head). If you want to execute a play against a server that's running a distribution other than Ubuntu, simply adjust `ansible.builtin.apt` to something else, such as `ansible.builtin.dnf`.

Since Ansible allows you to use additional resources from Galaxy, a site that's specific to Ansible that gives you access to additional capabilities, we use `ansible.builtin.apt` instead of simply `apt` in order to clarify that we're using a built-in module, not something we're using from an external resource.

This new naming scheme is new since the previous edition of this book was published. The older versions of the playbooks should still work just fine, but since the naming scheme has changed since then, we'll use the new naming style going forward.

You can, of course, add additional packages by simply adding more plays to the existing playbook:

```
---
- hosts: all
  become: true
  tasks:
    - name: Install htop
      ansible.builtin.apt:
        name: htop
    - name: Install git
      ansible.builtin.apt:
        name: git
    - name: Install vim-nox
      ansible.builtin.apt:
        name: vim-nox
```

However, that's not an extremely efficient method. I'll show you how we can combine multiple similar plays in one play. Sure, you don't have to, but I think you'll agree that this method looks cleaner:

```
---
- hosts: all
  become: true
  tasks:
    - name: Install packages
      ansible.builtin.apt:
        name:
          - git
          - htop
          - vim-nox
```

With the new format, we include just one play to install multiple packages. This is similar to the concept of a *for loop* if you have programming knowledge. For every package we list, it will run the `ansible.builtin.apt` module against it. If we want to add additional packages, we just add a new one to the list. Simple.

We can also copy files to our hosts. Consider the following example playbook, which I will call `copy_files.yml`:

```
---
- hosts: all
  become: true
  tasks:
    - name: copy SSH motd
      ansible.builtin.copy:
```

```
src: motd
dest: /etc/motd
```

Then, you can run this playbook with the following command:

```
ansible-playbook copy_files.yml
```

Inside the same directory, create a file called `motd` and place any text in it. It doesn't really matter what you type into the file, but this file in particular acts as a message that is printed any time a user logs into a server. When you run the playbook, it will copy that file over to the server at the destination you configured. Since we created a message of the day (`motd`), we should see the new message the next time we log in to the server.

By now, you're probably seeing just how useful Ansible can be. Sure, we only installed a few packages and copied one file. We could've performed those tasks easily ourselves without Ansible, but this is only a start. Ansible lets you automate everything, and we have to start somewhere. You can tell it to not only install packages and copy files, but you can also use it to start services, apply configuration file templates, and much more—it will surprise you. In fact, you can go as far as to automate the setup of a web server, a user's workstation... you name it!

Putting it all together – automating web server deployment

Speaking of automating the setup of a web server, why don't we go ahead and do exactly that? It'll be another simple example, but it will serve you well if we demonstrate more of what Ansible can do. We will set up a playbook to perform the following tasks:

1. Install Apache
2. Start the `apache2` service
3. Copy an HTML file for the new site

First, let's set up the playbook to simply install Apache. I called mine `apache.yml`, but the name is arbitrary:

```
---
- hosts: all
  become: true
  tasks:
    - name: Install Apache
      ansible.builtin.apt:
        name: apache2
```

No surprises here; we've already installed a package at this point. Let's add an additional instruction to start the `apache2` service:

```
---
- hosts: all
```

```

become: true
tasks:
- name: Install Apache
  ansible.builtin.apt:
    name: apache2
- name: Start the apache2 services
  ansible.builtin.service:
    name: apache2
    state: started

```

So far, the syntax should be self-explanatory. Ansible has a `service` module, which can be used to start a service on a host. In this case, we start the `apache2` service (the service would've already been started when `apache2` was installed, but at least this way we can make sure it's started). You do have to know the service name ahead of time, but you don't have to pay attention to what utility needs to be used in the background to start the service. Ansible already knows how to start services on all the popular distributions and takes care of the specifics for you in the background.

Having a play that starts `apache2` may seem a bit redundant since most packages you install on an Ubuntu server will automatically start the associated service as soon as it's installed (and `apache2` is no exception). But when we write automation code, it's important to be clear and explicit about what the desired end result is supposed to be. Even though `apache2` will automatically start once Ansible installs the package, we're adding a `service` play to clarify the fact that it needs to be running, so anyone looking at it will know what's expected. Also, we can make sure `apache2` is enabled by adding one more line to that section of the code (note the bold line):

```

- name: Start the apache2 services
  ansible.builtin.service:
    name: apache2
    state: started
    enabled: true

```

I'll leave it up to you whether or not to include that extra line, but the takeaway is to be as clear and direct as you can when writing automation, so there's no confusion about what the end result is supposed to be.

Well, that was easy. Let's create a simple web page for Apache to serve for us. It doesn't need to be fancy, we just want to see that it works. Create the following content inside a file named `index.html` in the same working directory as the other Ansible files:

```

<html>
<title>Ansible is awesome!</title>
<body>
  <p>Ansible is amazing. With just a small text file, we automated the setup
of a web server!</p>
</body>
</html>

```

As you can see, that HTML file is fairly lame, but it will work just fine for what we need. Next, let's add another instruction to our Apache playbook:

```
---
- hosts: all
  become: true
  tasks:
    - name: Install Apache
      ansible.builtin.apt:
        name: apache2
    - name: Start the apache2 services
      ansible.builtin.service:
        name: apache2
        state: started

    - name: Copy index.html
      ansible.builtin.copy:
        src: index.html
        dest: /var/www/html/index.html
```

With the copy module, we can copy a file from our local Ansible directory to the server. All we need to do is provide it with a source (src) and destination (dest).

Let's go ahead and run the new playbook:

```
ansible-playbook apache.yml
```

This will produce an output like the following:



```
ansible@ubuntu-server: ~
TASK [Install Apache] *****
changed: [10.10.10.222]

TASK [Start the apache2 services] *****
ok: [10.10.10.222]

TASK [Copy index.html] *****
changed: [10.10.10.222]

PLAY RECAP *****
10.10.10.222      : ok=4   changed=2   unreachable=0   failed=0   skipped=0
   rescued=0   ignored=0

ansible@ubuntu-server:~$
```

Figure 15.4: Installing Apache and copying a default site file with Ansible

Within a few minutes, you should have at least one web server configured by Ansible. In a real production environment, you would've only run this on servers within a specific role, but roles are beyond the scope of this chapter. But from the simple playbook we created, you should be able to see the power of this amazing piece of software:



Figure 15.5: An example of a web page provisioned by Ansible

At this point in our exploration into configuration management with Ansible, we've installed packages, started services, and copied files. Admittedly, this isn't much, but it's really all you need in order to practice the basics. The documentation for Ansible includes guides on how to do all sorts of things, and you'll be able to utilize the various modules it provides to perform many different tasks.

To explore Ansible further, I recommend that you think about things you do on a regular basis that you would benefit from automating. Things like installing security updates, creating user accounts, setting passwords, and enabling services to automatically start at boot time are tasks that are easy to get started with.

In addition, you may want to consider adding a simple **cron job** to run under your `ansible` user, to run the playbook every hour or so. We covered cron jobs in *Chapter 7, Controlling and Managing Processes*. Adding a cron job shouldn't add any overhead from a resource perspective, since Ansible won't actually be doing much unless you add a new command. In more advanced usage, you'll want to have Ansible check out code from a repository and then apply the configuration if it has changed. This way, all you have to do is commit to a Git repository and all of your servers will download the configuration and run it at the next scheduled time. One of the things I love most about Ansible is that it's easy to get started, but you'll continue to find new ways to use it and benefit from it.

Using Ansible's pull method

The way we set up our Ansible configuration in the previous section works very well if we have a list of specific servers that we want it to manage. To add a new server, we create the user account and set up the SSH configuration on the new host, and then add it to the inventory file. If we decommission that server, we simply remove it from the inventory file. This works well in a static environment, where servers you deploy typically stay around for a while. In a dynamic environment, though, this may not work as well.

Dynamic environments are very typical in the cloud. With cloud computing, you typically have one or more virtual servers that provide a service to your company or users. These servers may come and go at any time. With dynamic environments, servers will come online as needed to handle load and will also get decommissioned automatically as load decreases. Therefore, you never really know when a server is going to come online, and having to manually provision a server in such an environment is inefficient.

For this reason, Ansible's inventory file may not be a good fit for dynamic infrastructure. There certainly are ways to make Ansible's inventory work in such an environment, as you can actually replace the inventory file with an executable script that can make API calls and customize itself for your infrastructure if you so desire. However, that's out of the scope of this book, and there's an easier method anyway.

As we've seen, Ansible uses an inventory file and connects to every server listed in that file. However, Ansible also features a **pull mode**, where instead of having a central server that connects to other machines, each server in pull mode will actually run Ansible against itself. In my opinion, this is a great way to use Ansible and it doesn't seem to get the attention it deserves. First, I'll explain the theory of how it works, and then we can work through an actual example.

With pull mode, you'll want to have your Ansible playbooks inside a Git repository. This repository must be accessible from the servers you will manage. For example, if you store the Git repository on GitHub, you'll want to make sure the servers can access GitHub externally. If you host your own Git server internally, you'll want to make sure your servers are able to access it through your firewall or any security rules you may have in place.

Pull mode is used with the `ansible-pull` command, which comes bundled with Ansible. The syntax looks like the following:

```
ansible-pull -U https://github.com/myusername/ansible.git
```

Of course, you'd replace the URL with the actual HTTP or HTTPS URL to your Git repository. However, that's basically it. The `ansible-pull` command simply expects the `-U` option (short for URL) along with the URL to a Git repository.

In order for this to work, you'll need a playbook inside the repository with a special name, `local.yml`. If you don't declare a specific playbook with Ansible, it will expect to find a playbook with that name inside the root of the repository. If you choose to use a name for the main playbook that's something other than `local.yml`, you'll need to specify it:

```
ansible-pull -U https://github.com/myusername/ansible.git myplaybook.yml
```

In this example, the `ansible-pull` command will cache the Git repository located at the specified URL locally, and run the playbook `myplaybook.yml` that you would have inside the repository. One thing you may find is that Ansible might complain about not finding an inventory file, even though that's the entire point of the `ansible-pull` command. You can ignore this error. This will likely be fixed in Ansible at some point in the future, but as of the time of this writing, it will print a warning if it doesn't detect an inventory file.

With the theory out of the way, let's work through an actual example. If you've been following along so far, we created a playbook in the previous section that automates the deployment of a hypothetical web server. We can reuse that code. However, it's best practice to have a file with the name `local.yml`, so you can simply rename the `apache.yml` playbook we created earlier to `local.yml`. There's one small change we need to make to the file, which I've highlighted here:

```
---
- hosts: localhost
  become: true
  tasks:
    - name: Install Apache
      ansible.builtin.apt: name=apache2
    - name: Start the apache2 services
      ansible.builtin.service:
        name: apache2
        state: started
    - name: Copy index.html
      ansible.builtin.copy:
        src: index.html
        dest: /var/www/html/index.html
```

Since we're executing the playbook locally (without SSH) we changed the `hosts:` line to point to `localhost`, to instruct Ansible that we want to execute the commands locally rather than remotely. Now, we can push this playbook to our Git repository and execute it directly from the repository URL.

Pay careful attention to the `hosts:` line of any playbook you intend to run. If you are using the pull method, this line will need to be changed from `hosts: all` to `hosts: localhost`, the reason being, we are executing the playbooks directly on `localhost`, rather than from a remote SSH connection. If you don't make this change, you'll see an error similar to the following:

```
ERROR! Specified hosts and/or --limit does not match any hosts
```

Before you run the playbook, you'll want to first switch to your Ansible user, since the playbook will need to be run as a user with `sudo` privileges since it will execute system-level commands:

```
sudo su - ansible
```

Then, execute the playbook:

```
ansible-pull -U https://github.com/myusername/ansible.git
```

If we kept the name as `apache.yml`, we would just specify that:

```
ansible-pull -U https://github.com/myusername/ansible.git apache.yml
```

Keep in mind that since `ansible-pull` executes playbooks directly on localhost, the playbook must be executed by a user that has access to `sudo`. If `sudo` is not configured to run as that user without a password, the playbook will fail as it's not interactive (it won't ask for the password). You can also use `sudo` in front of the `ansible-pull` command and provide the password before it runs, but that won't work if you set it up to run automatically via `cron`.

If all goes according to plan, the playbook repository should be cached to the server, and the instructions carried out. If there is an error, Ansible is fairly good about presenting logical error messages. As long as the user has permission to execute privileged commands on the server, the required files are present in the repository, and the server has access to download the repository (a firewall isn't blocking it), then the playbook will run properly.

When it comes to implementing the pull method in production across various server types, there are several ways we can go about this. One way is to have a separate playbook per server type. For example, perhaps you'd have an Apache playbook, as well as playbooks specific to database servers, file servers, user workstations, and so on. Then, depending on the type of server you're deploying, you'd specify the appropriate playbook when you called the `ansible-pull` command. If you're using a service such as cloud computing, you can actually provide a script for each server to execute upon creation. You can then instruct the service to automatically run the `ansible-pull` command any time a new server is created. In AWS, for example, you can use a feature known as user data to place a script for a server to execute when it's launched for the first time. This saves you from having to provision anything manually. For this to work, you would first include a command for the server to install Ansible itself, and then the next command would be the `ansible-pull` command along with the URL to the repository. Just those two lines would completely automate the installation of Ansible and the application of your playbook. Try to think of the possibilities ahead of time so you can understand the many ways that automation can benefit you.

While provisioning new servers properly and efficiently is very important, so too is maintaining your existing servers. When you need to install a new package or apply a security update, the last thing you want to do is connect to all of your servers manually and update them one by one. The `ansible-pull` command allows simple management as well; you just simply run the command again. Every time you run `ansible-pull`, it will download the latest version of the code within your repository and run it. If there are any changes, they will be applied, while things that have already been applied will be skipped. For example, if you include a play to install the `apache2` package, Ansible won't reinstall the package if you run the playbook a second time; it will skip that play since that requirement has already been met.

One trick worth knowing with `ansible-pull` is the `-o` option. This option will ensure that the playbook inside the repository is only run if there have been any actual changes to the repository. If you haven't committed any changes to the repository, it will skip the entire thing. This is very useful if you set up the `ansible-pull` command syntax to be run periodically via `cron`, for example, every hour.

If you don't include the `-o` option, Ansible will run the entire playbook every hour. This will consume valuable CPU resources for no good reason at all. With the `-o` option, Ansible will only use as much CPU as required for simply checking the repository for changes. The playbook will only be run if you actually commit changes to the repository.

The introduction to Ansible within this chapter has been very basic, as we've only used the very core of the required components. By looking deeper into Ansible, you'll find more advanced techniques, and some more clever ways to implement it. Examples include automating firewall rules, security patches, and user passwords, as well as having Ansible send you an email any time a server is successfully provisioned (or even when that fails). Basically, just about anything you can do manually, you can automate with Ansible. In the future, I recommend looking at server administration with an automation mindset. As I've mentioned several times before, if you will need to perform a task more than once, automate it. Ansible is one of the best ways of automating server administration.

Summary

In this chapter, we took a look at configuration management using Ansible. Ansible is an exciting technology that is exploding in popularity. It gives you the full power of configuration management utilities such as Chef or Puppet, without all the resource overhead. It allows you to automate just about everything. During our exploration, we walked through installing packages, copying files, and starting services. Near the end of the chapter, we worked through an example of using Ansible to provision a simple web server, and we even explored the pull method, which is very useful in dynamic environments. These concepts form the basis of knowledge that can be expanded to automate more complex rollouts.

The next chapter will be fun: we'll set up our very own virtualization server with KVM. This is one of my favorite topics, and I'm sure you'll enjoy it too. See you there!

Relevant videos

- Git Essentials (LearnLinuxTV): <https://linux.video/git-essentials>
- Getting Started with Ansible (LearnLinuxTV): <https://linux.video/learn-ansible>
- Ansible workstation/laptop configuration (LearnLinuxTV): <https://linux.video/ansible-workstation>
- Ansible Vault (LearnLinuxTV): <https://linux.video/ansible-vault>
- Ansible Pull (LearnLinuxTV): <https://linux.video/ansible-pull>

Further reading

- Ansible documentation: <https://learnlinux.link/ansible-docs>
- Ansible Roles documentation: <https://learnlinux.link/reuse-roles>
- Ansible config file documentation article: <https://learnlinux.link/a-config>
- ansible-pull documentation: <https://learnlinux.link/a-pull>
- How to manage your workstation configuration with Ansible: <https://learnlinux.link/a-ws>
- Git basics: <https://learnlinux.link/git-book>
- Setting up Git (GitHub): <https://learnlinux.link/setup-git>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



16

Virtualization

There have been a great many advancements in the IT space in the last few decades, and a few technologies have come along that have truly revolutionized the technology industry. I'm sure few would argue that the internet is by far the most revolutionary technology to come around, but another technology that has created a paradigm shift in IT is virtualization. This concept changed the way we maintain our data centers, allowing us to segregate workloads into many smaller machines being run from a single server. This allows us to get even more use out of our hardware. Since Ubuntu features the latest advancements of the Linux kernel, virtualization support is actually built right into it. After installing just a few packages to allow us to interact with the virtualization features, we can create virtual machines on our Ubuntu server without the need for a pricey license agreement or support contract. In this chapter, I'll walk you through setting up your own Ubuntu-based virtualization solution. Along the way, I'll walk you through the following topics:

- Prerequisites and considerations
- Setting up a virtual machine server
- Creating virtual machines
- Bridging the virtual machine network
- Simplifying virtual machine creation with cloning
- Managing virtual machines via the command line

In order to get started, we'll need a server to use for this task, and we'll first have a discussion on some considerations to make when setting up a server for this purpose.

Prerequisites and considerations

I'm sure many of you have already used a virtualization solution before. In fact, I bet a great many readers are following along with this book while using a **Virtual Machine (VM)** running in a solution such as VirtualBox, Parallels, VMware, or one of the others. Those applications and others like them are great for testing Ubuntu or other operating systems on your desktop or laptop. In this section, we'll set up a VM server that can act as a centrally available server on which to run VMs.

This will be easier than you may think—Ubuntu has virtualization built right in. This comes in the form of a dynamic duo consisting of **Kernel-based VM (KVM)** and **Quick Emulator (QEMU)**, which together form a virtualization suite that enables Ubuntu (and Linux in general) to run VMs without the need for a third-party solution. KVM is the feature that is built right into the Linux kernel that performs the magic under the hood. It handles the low-level instructions in the kernel that are needed to separate tasks between those run on a physical host and on a guest VM. QEMU is also important, as it emulates hardware components that are generally found in physical servers. The combination of KVM and QEMU makes up the virtualization solution that can be enabled on an Ubuntu server to turn it into a host for VMs.

To be fair, you can set up something like VirtualBox on your Ubuntu Server to accomplish the same thing, and end up with a centrally available virtualization server. And that's perfectly valid, there's certainly nothing wrong with running VirtualBox this way, and many people do. But there are improvements to be had by utilizing a built-in system, and KVM offers a very fast interface to the Linux kernel to run your VMs with near-native speeds, depending on your use case. QEMU/KVM (which I'll refer to simply as KVM going forward) is about as native as you can get.

I bet you're eager to get started, but there are a few quick things to consider before we dive in. First, of all the activities I've walked you through in this book so far, setting up our own virtualization solution will be the most expensive from a hardware perspective. The more VMs you plan on running, the more resources your server will need to have available (especially RAM). Thankfully, most computers nowadays ship with 8 GB of RAM at a minimum, with 16 GB or more being fairly common. With most modern computers, you should be able to run VMs without too much of an impact. Depending on what kind of machine you're using, the CPU and RAM may present bottlenecks, especially when it comes to legacy hardware.

For the purposes of this chapter, it's recommended that you have a PC or server available with a processor that's capable of supporting VM extensions. A good majority of CPUs on computers nowadays offer this, though some may not. To be sure, you can run the following command on the machine you intend to host the KVM VMs on in order to find out whether your CPU supports virtualization extensions:

```
egrep -c '(vmx|svm)' /proc/cpuinfo
```

A result of 1 or more means that your CPU does support virtualization extensions. A result of 0 means it does not:



```
jay@ubuntu-server: ~  
jay@ubuntu-server:~$ egrep -c '(vmx|svm)' /proc/cpuinfo  
8  
jay@ubuntu-server:~$
```

Figure 16.1: Checking the CPU for compatibility with virtualization

Even if your CPU does support virtualization extensions, it's often the case that it's disabled by default with most end user PCs sold today, and even some servers. To enable these extensions, you may need to enter the BIOS setup screen for your computer and enable the option. Depending on your CPU and chipset, this option may be named something similar to “virtualization support,” under a more technical name such as VT-x, AMD-V, or another verbiage. Unfortunately, I won't be able to walk you through how to enable the virtualization extensions for your hardware, since the instructions will differ from one machine to another. If in doubt, refer to the documentation for your hardware.

One final note: I'm sure many of you are using VirtualBox, as it seems to be a very popular solution for those testing out Linux distributions (and rightfully so; it's great!). However, you can't run both VirtualBox and KVM VMs on the same machine simultaneously. You can certainly have both solutions installed on the same machine, but you just can't have a VirtualBox VM up and running, and then expect to also be able to start up a KVM VM. The virtualization extensions of your CPU can only work with one solution at a time.

Another consideration to bear in mind is the amount of space the server has available, as VMs can take quite a bit of space. The default directory for KVM VM images is `/var/lib/libvirt/images`. If your `/var` directory is part of the root filesystem, you may not have a lot of space to work with here. One trick is that you can mount an external storage volume to this directory, so you can store your VM disk images on another volume. Or you can simply create a symbolic link that will point this directory somewhere else. We discussed symbolic links in *Chapter 5, Managing Files and Directories*. The choice is yours. If your root filesystem has at least 10 GB available, you should be able to create at least one VM without needing to configure the storage. I think it's a fair estimate to assume at least 10 GB of hard drive space per VM.

Setting up a virtual machine server

With all the discussion out of the way, let's start the process and set up our virtualization server. Even though KVM is built into the Linux kernel, we'll still need to install some packages in order to properly interface with it. Specifically, we'll need to install several `libvirt` packages, as well as `QEMU` itself. `libvirt` itself gives us access to manage virtualization platforms on our server, as it provides us with a set of useful tools to manage our virtual machines.

These packages will require a decent number of dependencies, so it may take a few minutes for everything to install:

```
sudo apt install bridge-utils libvirt-clients libvirt-daemon-system qemu-system-x86
```

You'll now have an additional service running on your server, `libvirtd`. Once you've finished installing KVM's packages, this service will be started and enabled for you. Feel free to take a look at it to see for yourself:

```
systemctl status libvirtd
```

You should see information on the state of the service, similar to the following:

```

jay@ubuntu-server:~$ systemctl status libvirtd.service
● libvirtd.service - Virtualization daemon
   Loaded: loaded (/lib/systemd/system/libvirtd.service; enabled; vendor preset: enabled)
   Active: active (running) since Mon 2022-05-23 16:14:00 EDT; 32s ago
     TriggeredBy: ● libvirtd-admin.socket
                  ● libvirtd-ro.socket
                  ● libvirtd.socket
   Docs: man:libvirtd(8)
          https://libvirt.org
   Main PID: 2934 (libvirtd)
     Tasks: 21 (limit: 32768)
    Memory: 9.6M
      CPU: 484ms
   CGroup: /system.slice/libvirtd.service
           └─2934 /usr/sbin/libvirtd
             └─3064 /usr/sbin/dnsmasq --conf-file=/var/lib/libvirt/dnsmasq/default.conf --
             └─3065 /usr/sbin/dnsmasq --conf-file=/var/lib/libvirt/dnsmasq/default.conf --
lines 1-16

```

Figure 16.2: Checking the status of the `libvirtd` unit after installing KVM-related packages

Let's stop this service for now, as we have some additional configuration to do:

```
sudo systemctl stop libvirtd
```

Next, we'll need to make sure we have two required groups on our server, `kvm` and `libvirt`. It's quite possible that the packages that we've installed have added these groups on our server already, so feel free to check the contents of `/etc/group` and see if they're there. If not, you can create them with the `groupadd` command:

```
sudo groupadd kvm
sudo groupadd libvirt
```

Our primary user account should be a member of both groups. If your user isn't already a member of these, add your user to the required groups (substitute the username, `jay`, with yours):

```
sudo usermod -aG kvm jay
sudo usermod -aG libvirt jay
```

At this point, you may as well log out and then log in again to ensure the changes to your group memberships have taken effect.

To ensure we'll be able to manage virtualization properly, we should ensure that users of the `kvm` group have access to the `/var/lib/libvirt/images` directory so that they'll have access to the data that will be stored in the directory. First, we'll apply the `kvm` group to this folder:

```
sudo chown :kvm /var/lib/libvirt/images
```

Then, we'll set the permissions of `/var/lib/libvirt/images` such that anyone in the `kvm` group will be able to modify its contents:

```
sudo chmod g+rw /var/lib/libvirt/images
```

With the initial packages and permissions in place, we can now start the `libvirtd` service:

```
sudo systemctl start libvirtd
```

Next, check the status of the service to make sure that there are no errors:

```
sudo systemctl status libvirtd
```

Now that we've configured the server, we can set up our workstation to be able to connect to it and manage the virtualization implementation that we've set up. We'll install a utility that will give us a **graphical user interface** (GUI) through which we can perform administration tasks relating to VMs. The utility we'll be using for this purpose is known as **Virtual Machine Manager** abbreviated as `virt-manager`. This utility is installed on Linux workstations, so you'll need to install it on a laptop or desktop that's running a desktop variant of Linux. If you have a computer running Debian or Ubuntu, the following command will install the packages that are required for this:

```
sudo apt install ssh-askpass virt-manager
```

If you use a distribution of Linux other than Ubuntu, Debian, or one based on them, then you may need to consult the documentation for your distribution in order to install `virt-manager`. If you're not running Linux on your workstation at all, there is a suite of command-line utilities that can be used to manage VMs that we'll cover later in this chapter when we discuss this in the *Managing virtual machines via the command line* section. If all else fails, you can install this utility inside a Linux VM running on your workstation.

Next, open `virt-manager` on your administration machine. It should be located in the **Applications** menu of your desktop environment, usually under the **System Tools** section of **Virtual Machine Manager**. If you have trouble finding it, simply run `virt-manager` in your shell prompt. When you first launch it, you may see the following error:

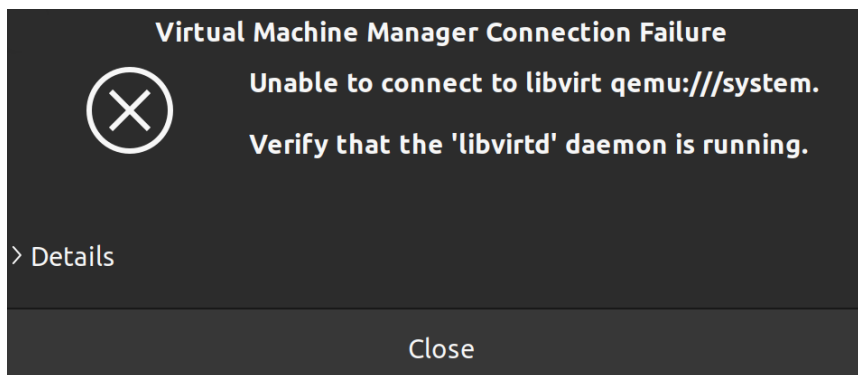


Figure 16.3: A possible error that may appear when first launching `virt-manager`

If you do see the error, simply dismiss it and don't worry about it. By default, `virt-manager` defaults to attempting to connect to an instance of `libvirtd` running on your local computer. Unless you are also running KVM VMs locally and you've already set it up, this attempt will fail. But that doesn't matter for us, as we'll be using `virt-manager` to manage VMs on our server.

Once you've opened `virt-manager`, you'll see the main window, which will look similar to the following:

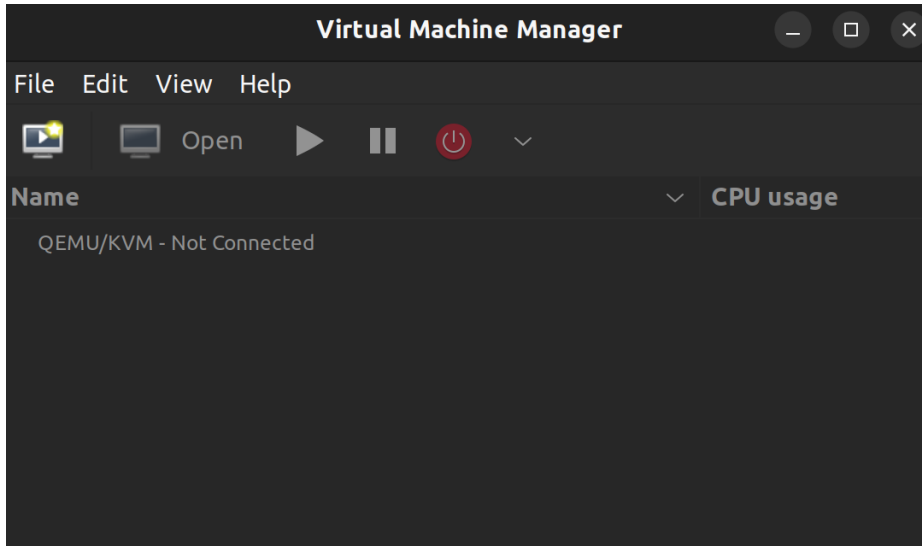


Figure 16.4: The `virt-manager` application

The `virt-manager` utility is especially useful as it allows us to manage both remote and local KVM servers. From one utility, you can create connections to any of your KVM servers, including one or more external servers or `localhost` if you are running KVM on your laptop or desktop. To create a new connection, click on **File** and select **Add Connection**. A new screen will appear, where we can fill out the details of the KVM server we wish to connect to:

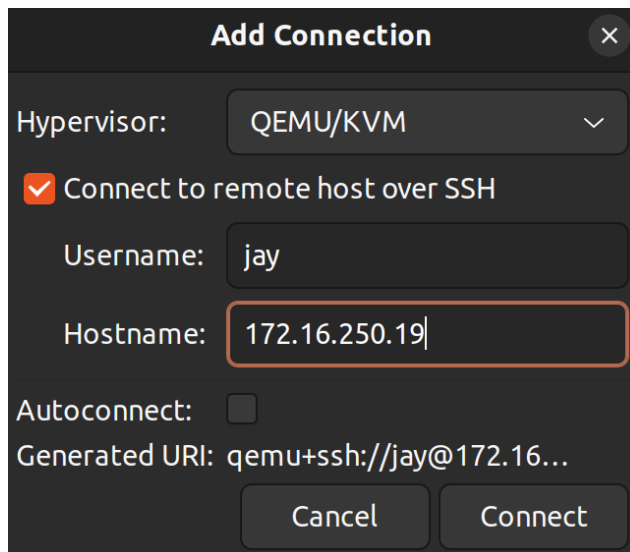


Figure 16.5: Adding a new connection to `virt-manager`

In the **Add Connection** window, enter the details of your connection. In the screenshot, you can see that I first checked the **Connect to remote host over SSH** box, which selects SSH as my connection method, `jay` for my **Username**, and I've entered the IP address of my KVM server (`172.16.250.19`) in the **Hostname** field. Fill out the specific details here for your KVM server to set up your connection. Keep in mind that in order for this to work, the username you include here will need to be able to access the server via SSH and have permissions to the hypervisor (be a member of the `kvm` and `libvirtd` groups we added earlier), and the `libvirtd` service must be running on the server. If all of these requirements are met, you'll have a new connection set up to your KVM server when you click **Connect**. You might see a pop-up dialog box with the text `Are you sure you wish to continue connecting (yes/no)?`. If you do, type `yes` and press *Enter*.

Either way, you should be prompted for your password to your KVM server; type that in and press *Enter*. You should now have a connection listed in your `virt-manager` application. You can see the connection I added in the following screenshot; it's the second one on the list. The first connection is `localhost` since I also have KVM running on my local laptop in addition to having it installed on a remote server:

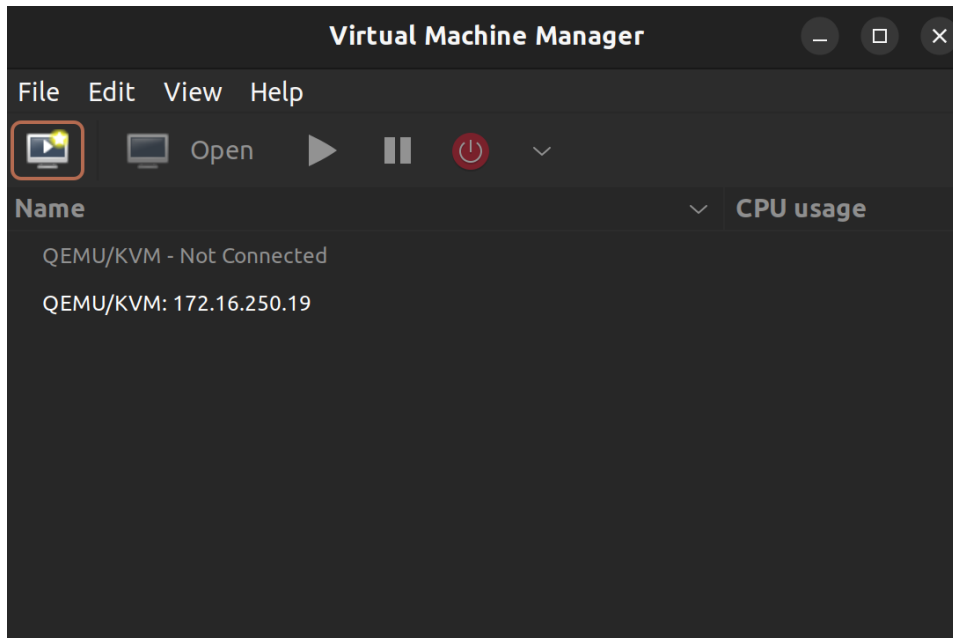


Figure 16.6: *virt-manager* with a new connection added

We're almost at a point where we'll be able to test our KVM server. But first, we'll need a storage group for ISO images, for use when installing operating systems on our VMs. When we create a VM, we can attach an ISO image from our ISO storage group to our VM, which will allow it to install the operating system.

To create this storage group, open virt-manager if it's not open already. Right-click on the listing for your server connection, and then click on **Details**. You'll see a new window that will show details regarding your KVM server. Click on the **Storage** tab:

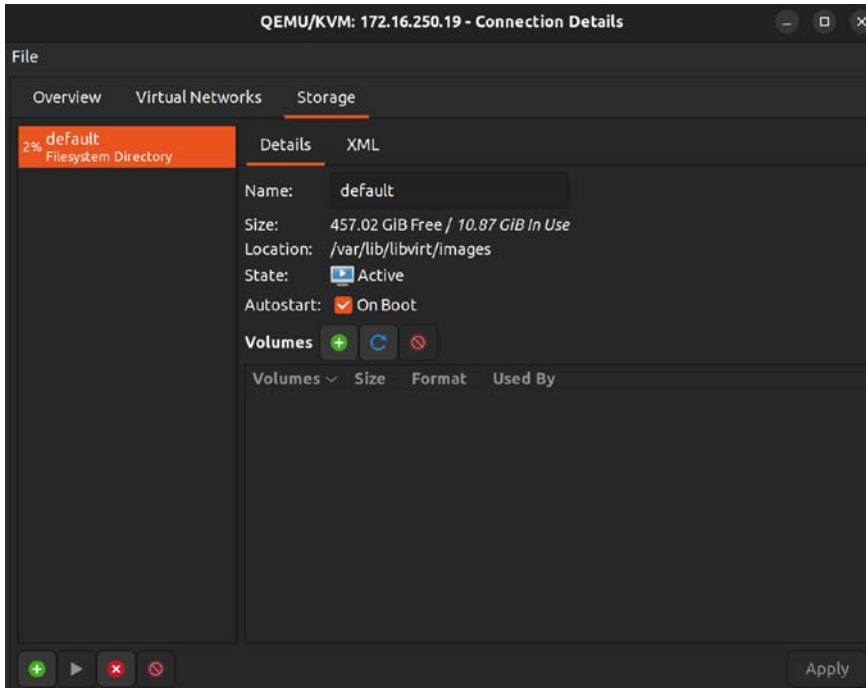


Figure 16.7: The first screen while setting up a new storage pool

At first, you'll only see the default connection we edited earlier. Now, we can add our ISO storage pool. Click on the plus symbol in the bottom-left corner to create the new pool:

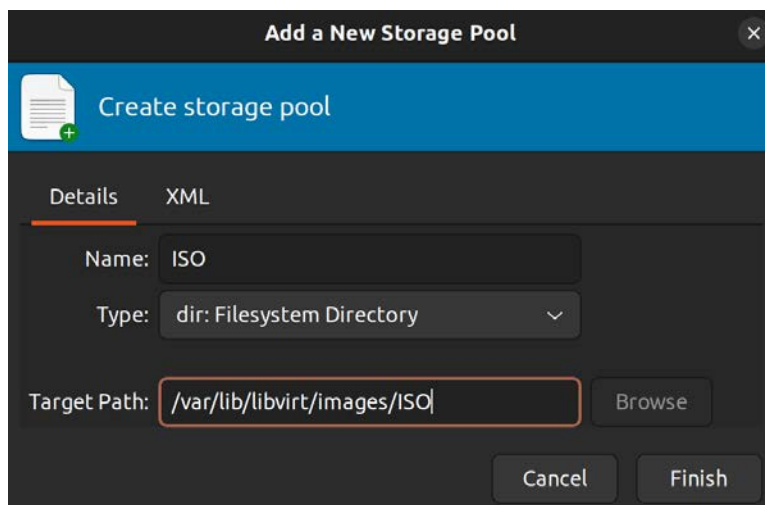


Figure 16.8: The storage tab of the virt-manager application

In the **Name** field, type `ISO`. You can actually name it anything you want, but `ISO` makes sense, considering it will be storing ISO images. For the **Target Path** field, set it to `/var/lib/libvirt/images/ISO` unless you have a different directory in your filesystem for VM storage. Click **Finish** to finalize our changes. We should also update the permissions for this directory so that it's owned by the proper user, and members of the `kvm` group have read and write access to it:

```
sudo chown root:kvm /var/lib/libvirt/images/ISO
sudo chmod g+rw /var/lib/libvirt/images/ISO
```

Congratulations! You now have a fully configured KVM server for creating and managing VMs. Our server has a place to store VMs as well as ISO images. You should also be able to connect to this instance using `virt-manager`, as we've done in this section. Next, I'll walk you through the process of setting up your first VM. Before we get to that, I recommend you copy some ISO images over to your KVM server. It doesn't really matter which ISO image you use—any operating system should suffice. If in doubt, you can simply download Ubuntu Server 22.04 again like we did back in *Chapter 1, Deploying Ubuntu Server*, when we set up our initial installation.

After you've chosen an ISO file and you've downloaded it, copy it over to your server via `scp` or `rsync`, and move it into the `/var/lib/libvirt/images/ISO` directory. Both of those utilities were covered in *Chapter 12, Sharing and Transferring Files*. Once the file has been copied over, you should have everything you need for now.

Creating virtual machines

Now, the time has come to put your new VM server to the test and create a VM. At this point, I'm assuming that the following is true:

- You're able to connect to your KVM server via `virt-manager`
- You've already copied one or more ISO images to the server
- Your storage directory has at least 10 GB of space available
- The KVM server has enough free RAM to be associated with the VM you intend on creating

Go ahead and open up `virt-manager`, and let's get started!

In `virt-manager`, right-click your server connection and click on **New** to start the process of creating a new VM. The default selection will be on **Local install media (ISO image or CDROM)**; leave this selection as is and click on **Forward**:

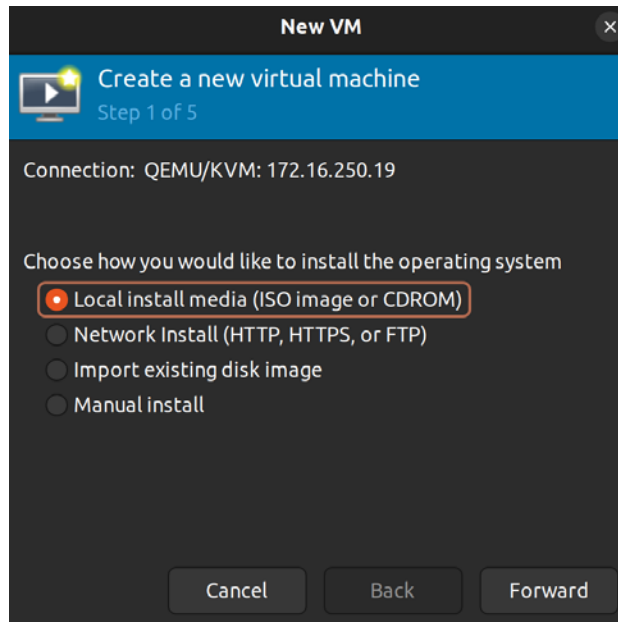


Figure 16.9: The first screen while setting up a new VM

On the next screen, click on **Browse** to open up another window where you can select the ISO image you've downloaded:

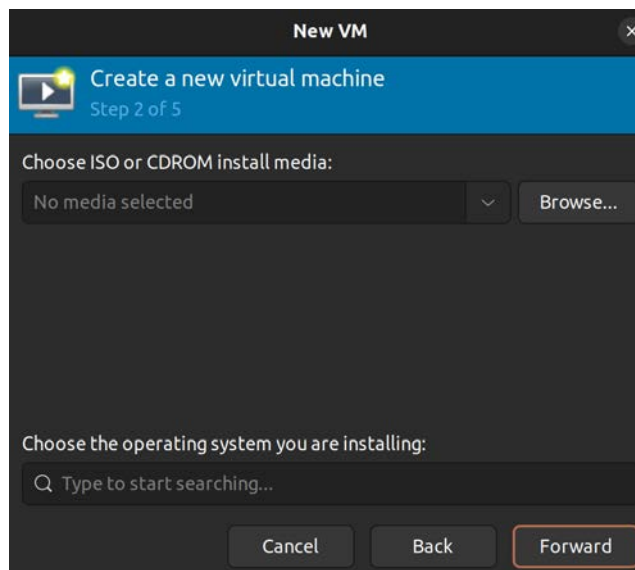


Figure 16.10: Creating a new VM and setting the VM options

If you click on your ISO storage pool, you should see a list of ISO images you've downloaded:

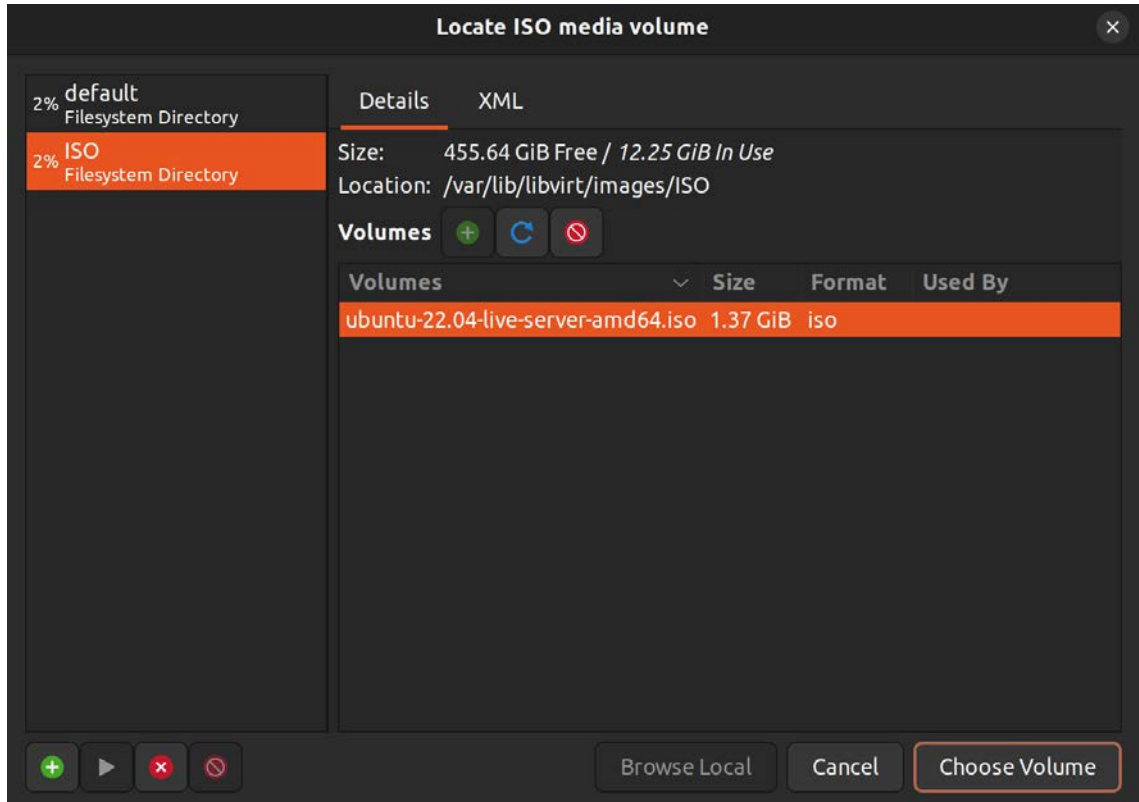


Figure 16.11: Choosing an ISO image during VM creation

If you don't see any ISO images here, you may need to click the refresh icon. In my sample server, I added an install image for Ubuntu Server 22.04, which you can see in the list. Again, you can use whatever operating system you prefer. Click on the ISO image name to highlight it, and then click **Choose Volume** to finalize the selection. Then, click **Forward** to continue to the next screen.

Next, you'll be asked to allocate RAM and CPU resources to the VM:

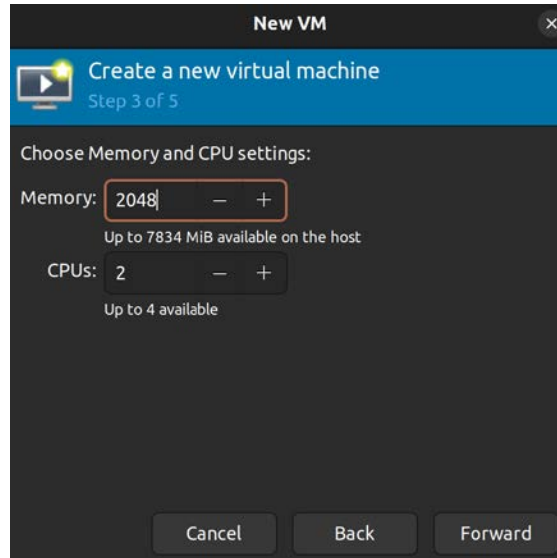


Figure 16.12: Adjusting the RAM and CPU count for the new VM

For most Linux distributions with no GUI, 2,048 MB is plenty (unless your workload demands more). One CPU core is fine for lightweight workloads, but consider adding more if the documentation for the application you intend on running recommends more than that. The resources you select here will depend on what you have available on your host. Click on **Forward** when you've finished allocating resources.

Next, you'll allocate free disk space for your VM's virtual hard disk:

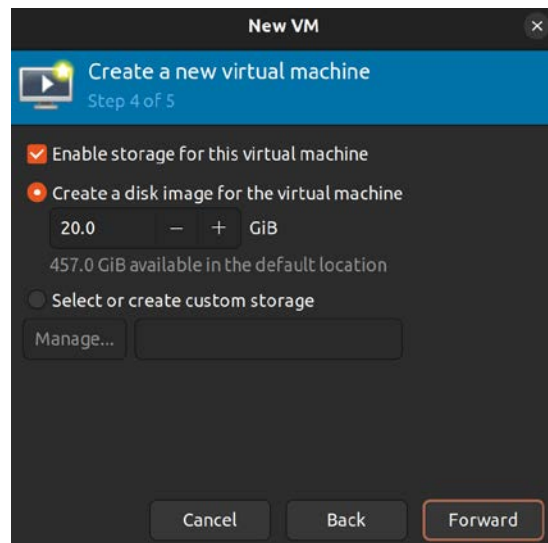


Figure 16.13: Allocating storage resources for the new VM

Set the disk image size to however much space you feel is relevant for the purpose of the VM. Click on **Forward** when done.

Finally, you'll name your VM:

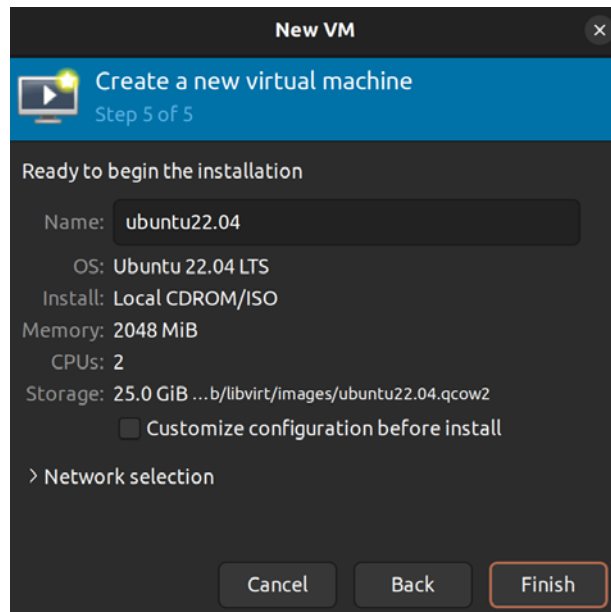


Figure 16.14: Naming the new VM

This won't be the hostname of the VM; it's just the name you'll see when you see the VM listed in `virt-manager`. When you click on **Finish**, the VM will start and it will automatically boot into the install ISO you've attached to the VM near the beginning of the process. The installation process for that operating system will then begin:

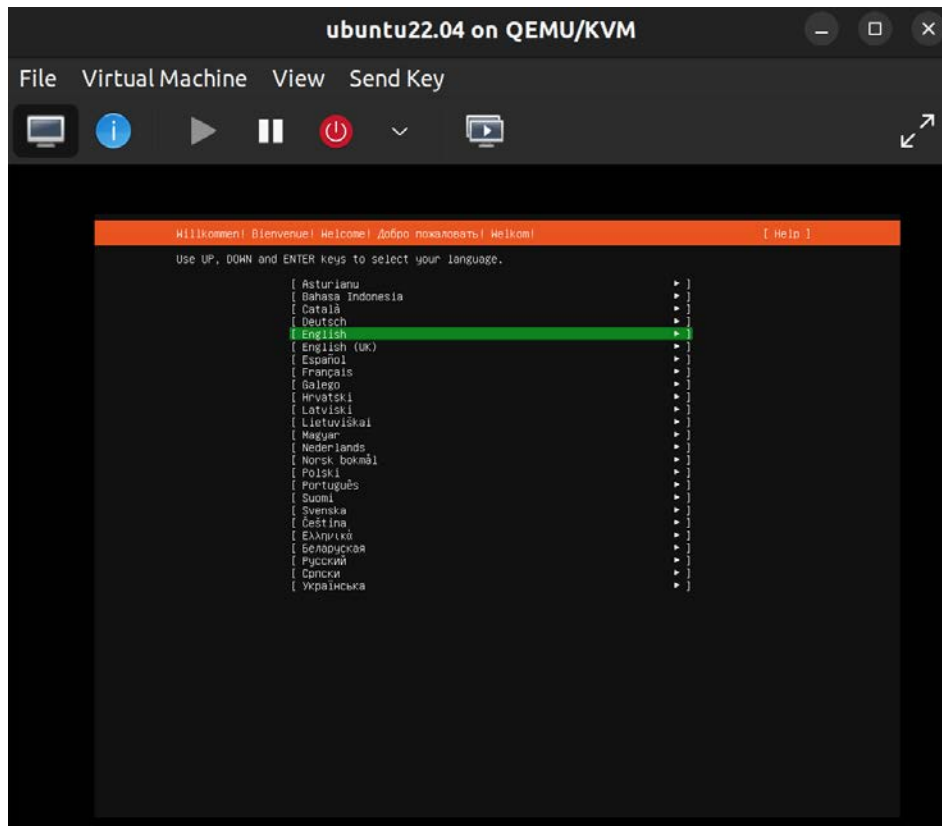


Figure 16.15: Installing Ubuntu Server inside a VM

When you click on the VM window, it will steal your keyboard and mouse and dedicate them to the window. Press `Ctrl` and `Alt` at the same time to release this control and regain full control of your keyboard and mouse.

Unfortunately, I can't walk you through the installation process of your VM's operating system since there are hundreds of possible candidates you may be installing. If you're installing another instance of Ubuntu Server, you can refer back to *Chapter 1, Deploying Ubuntu Server*, where we walked through the process. The process will be the same in the VM. From here, you should be able to create as many VMs as you need and have resources for.

Next, we'll look at some concepts surrounding networking for our VMs.

Bridging the virtual machine network

Your KVM VMs will use their own network unless you configure bridged networking. This means your VMs will get an IP address in their own network, instead of yours. By default, each machine will be a member of the `192.168.122.0/24` network, with an IP address in the range of `192.168.122.2` to `192.168.122.254`. If you're utilizing KVM VMs on your personal laptop or desktop, this behavior might be adequate. You'll be able to SSH into your VMs via their IP addresses if you're connecting from the same machine the VMs are running on. If this satisfies your use case, there's no further configuration you'll need to do.

Bridged networking allows your VMs to receive an IP address from the DHCP server on your network instead of its internal one, which will allow you to communicate with your VMs from any other machine on your network. This use case is preferable if you're setting up a central VM server to power infrastructure for your small office or organization, as your DHCP server can become a single source of truth for all of the IP addresses in use in your organization. With a bridged network on your VM server, each VM will be treated as any other network device. All you'll need is a wired network interface, as wireless cards typically don't work with bridged networking.

That last point is very important. Some network cards don't support bridging, and if yours doesn't, you won't be able to use a bridge with your VM server unless you replace the network card. Before continuing, you may want to ensure your network card supports bridging by reading the documentation from the vendor of your device. In my experience, most wired cards made by Intel support bridging, and most wireless cards do NOT. Make sure you back up the Netplan configuration file before changing it, so you can revert back to the original version if you find that bridging doesn't work for you.

To set up bridged networking, we'll need to create a new interface on our server (the one that's intended for hosting virtual machines). Open up the `/etc/netplan/00-installer-config.yaml` file in your text editor with `sudo`. We already talked about this file in *Chapter 10, Connecting to Networks*, so I won't go into too much detail about it here. Basically, this file includes the configuration for each of our network interfaces, and this is where we'll add our new bridged interface.

Make sure you make a backup of the original Netplan configuration file, and then replace its contents with the following. Be sure to replace `enp0s3` (the interface name) with your actual wired interface name if it's different. There are two occurrences of it in the file.

If you're reading the digital version of this book, it's highly recommended that you refrain from copying and pasting the following code, but rather type it manually or copy it from the GitHub URL for the book's code bundle. The reason is that the YAML format is extremely picky about spaces, and if you end up with a mix of spaces and tabs, the file might not work. When Netplan errors, it can be very hard to figure out exactly what it's complaining about, but spacing is quite often the culprit even if the error output doesn't lead you to believe so.

Take your time while configuring this file. If you make a single mistake, you will likely not have network access to the machine once it restarts:

```
network:
  ethernets:
    enp0s3:
      dhcp4: false
  bridges:
    br0:
      interfaces: [enp0s3]
      dhcp4: true
      parameters:
        stp: false
        forward-delay: 0
```

After you make the change, you can apply the new settings immediately, or simply reboot the server. If you have a monitor and keyboard hooked up to the server, the following command is the easiest way to activate the new configuration:

```
sudo netplan apply
```

If you're connected to the server via SSH, restarting the network configuration will likely result in the server becoming inaccessible because the SSH connection will likely drop as soon as the network stops. This will disrupt the connection and prevent networking from starting back up. If you know how to use `screen` or `tmux`, you can run the `restart` command from within either; otherwise, it may just be simpler for you to reboot the server.

After networking restarts or the server reboots, check whether you can still access network resources, such as pinging websites and accessing other network nodes from it. If you can, you're all set. If you're having any trouble, make sure you edited the Netplan config file properly.

Now, you should see an additional network interface listed when you run `ip addr show`. The interface will be called `br0`. The `br0` interface should have an IP address from your DHCP server, in place of your `enp0s3` interface (or whatever it may be named on your system). From this point forward, you'll be able to use `br0` for your VM's networking, instead of the internal network. The internal KVM network will still be available, but you can select `br0` to be used instead when you create new VMs.

If you have a VM you've already created that you'd like to switch to utilize your bridged networking, you can use the following steps to convert it:

1. First, open `virt-manager` and double-click on your VM. A new window with a graphical console of your VM will open.
2. The second button along the top (which appears as a blue circle) will open the **Virtual Hardware Details** tab, which will allow you to configure many different settings for the VM, such as the CPU count, the RAM amount, the boot device order, and more.

3. Among the options on the left-hand side of the screen, there will be one that reads NIC and shows part of the VM's network card's MAC address. If you click on this, you can configure the VM to use your new bridge by selecting it in the list.
4. Finally, click on **Apply**. You may have to restart the VM for the changes to take effect:

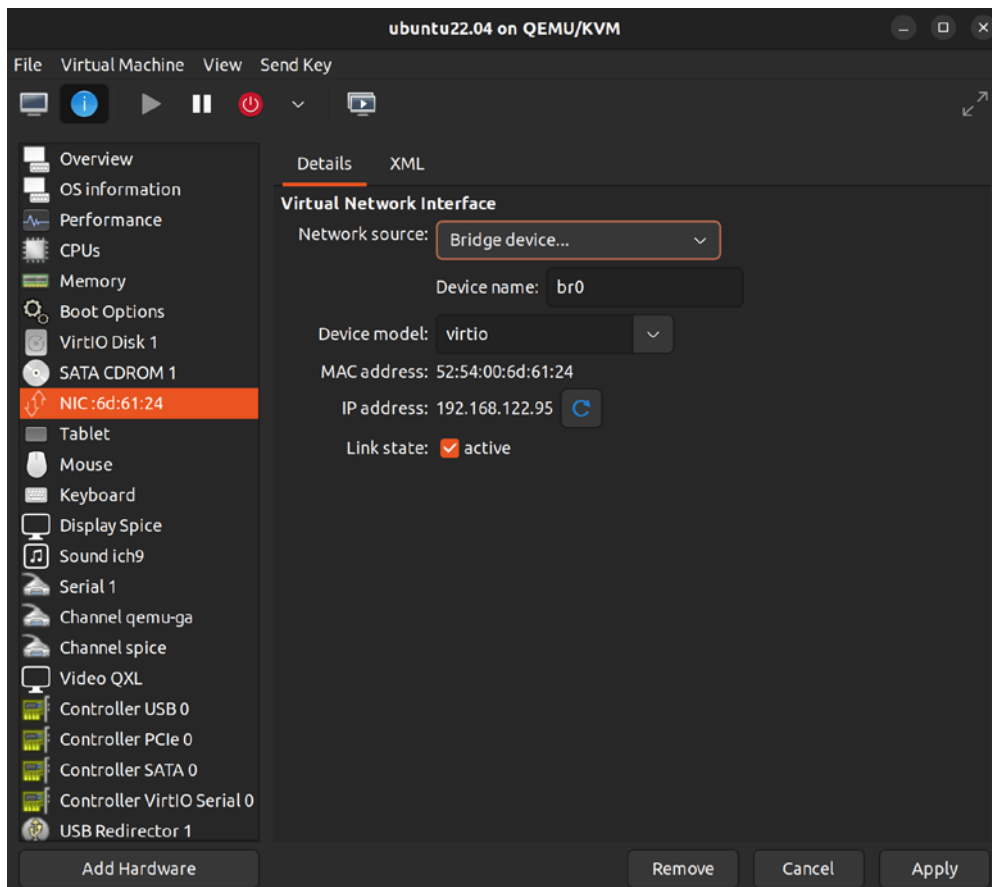


Figure 16.16: Configuring a VM to use bridge `br0`

While creating a brand-new VM, there's an additional step you'll need to do in order to configure the VM to use bridged networking. In the last step of the process, where you set a name for the VM (as shown in *Figure 16.14*), you'll also see **Advanced options** listed near the bottom of the window. Expand this, and you'll be able to set your network name. Change the dropdown in this section to **Specify shared device name** and set the bridge **Name** to `br0`. Now, you can click on **Finish** to finalize the VM as before, and it should use your bridge whenever it starts up.

From this point onward, you should have not only a fully configured KVM server or instance but also a solution that can be treated as a full citizen of your network. Your VMs will be able to receive an IP address from a DHCP server and communicate with other network nodes directly. If you have a very beefy KVM server, you may even be able to consolidate other network appliances into VMs to save space, which is basically the entire purpose of virtualization.

In the next section, we'll simplify the process a bit by discussing the creation of a template that can be used to act as a preconfigured starting point when setting up a new VM.

Simplifying virtual machine creation with cloning

Now that we have a KVM server, and we can spin up an army of VMs to do our bidding, we can try and find clever ways of automating some of the workload of setting up a new VM. Every time we go to create a new VM, we need to go through the entire installation process for its operating system again. While this process is not difficult, we can certainly simplify it.

Most prominent virtualization solutions include a feature that allows you to create a **VM Template**. With a template, we can create a VM once and get it completely configured. Then, we can convert it into a template and use it as a base for all future VMs that will use that same operating system. This saves a tremendous amount of time. You'll probably recall the handful of screens you had to navigate through to install Ubuntu Server in our first chapter. Imagine not having to go through that process again (or at least not nearly as often).

Unfortunately, as great as QEMU/KVM is, it doesn't have a template feature. This glaring hole in its feature set is a sizable setback, but thankfully we Linux administrators are very clever, and we can easily work around this to create a solution that's essentially the same thing as templates.

Take the following screenshot, for example:

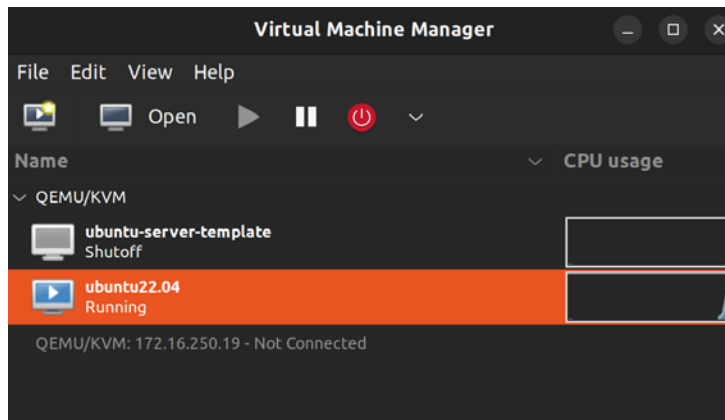


Figure 16.17: Virtual Machine Manager with a template listed

In the screenshot, you can see two VMs, `ubuntu22.04` and `ubuntu-server-template`. Although its name would lead you to believe otherwise, the latter is not a template at all; it's just a VM. There's nothing really different about it, aside from the fact that it isn't running. What it is, though, is a clever workaround (if I do say so myself). If I want to create a new VM, I simply right-click on it, then click **Clone**.

The following window will appear:

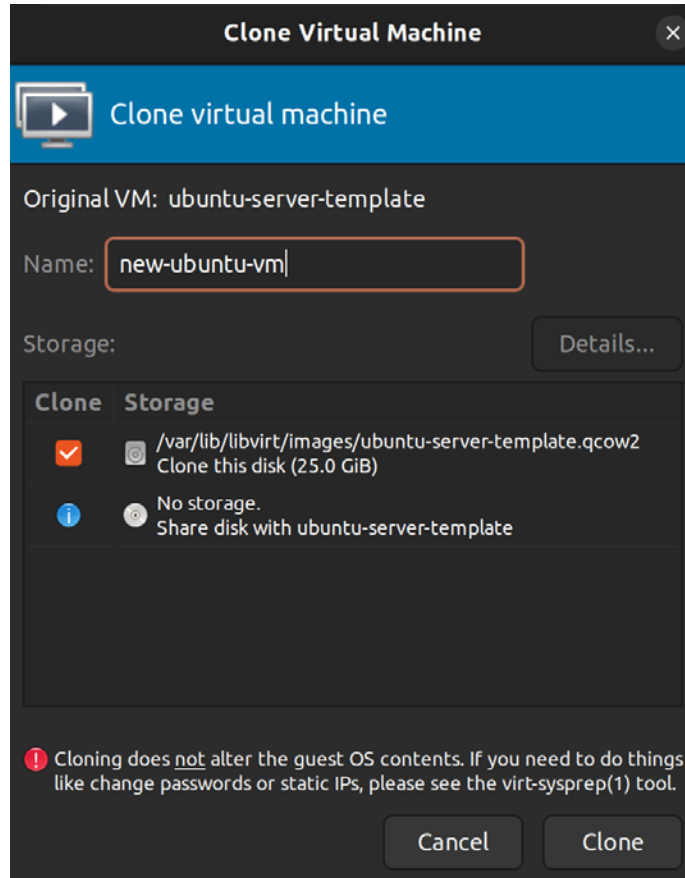


Figure 16.18: Cloning a VM



When I click **Clone** in this window, after giving the new VM a name, I've made a copy of it to serve as my new VM. It will use the original as a base, which I've already configured. Since Ubuntu Server was installed on the "template," I don't need to do all that work again.

If you create virtual machine templates for production use, it's highly recommended that you check out **cloud-init**, which can help generalize its Ubuntu installation, which includes regenerating its SSH host keys and the machine ID. cloud-init is beyond the scope of this book but is definitely essential if you want to go even deeper into the topic of generating virtual machine templates.

Think about the tasks that you find yourself doing manually after setting up a new Ubuntu Server instance. With a base VM being used as if it were a template, you can include any tweaks or customizations you find yourself implementing right into that VM, so every time you clone it, all that work is done for you automatically. So long as you maintain your base VM, you can spin up as many VMs from it as you need and be able to do so with minimal configuration steps.

We've used `virt-manager` quite a bit in this chapter to customize our VMs, and while it's a great utility, we should also understand how to manage our infrastructure without it. In the next section, we'll take a look at some command-line examples of managing VMs.

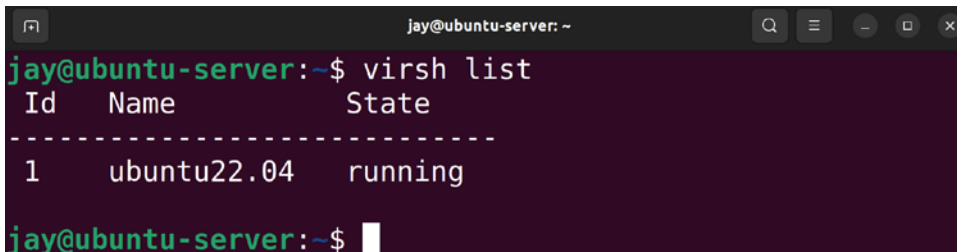
Managing virtual machines via the command line

In this chapter, I showed you how to manage VMs with `virt-manager`. This is great if you have a secondary machine with a GUI running Linux as its operating system. But what do you do if such a machine isn't available, and you'd like to perform simple tasks such as rebooting a VM or checking to see which VMs are running on the server?

On the VM server itself, you have access to the `virsh` suite of commands, which will allow you to manage VMs even if a GUI isn't available. To use these commands, simply connect to the machine that stores your VMs via SSH. What follows are some easy examples to get you started. Here's the first one:

```
virsh list
```

This command will return an output like that shown in the following screenshot:



```
jay@ubuntu-server:~$ virsh list
Id   Name           State
-----
 1   ubuntu22.04    running
jay@ubuntu-server:~$
```

Figure 16.19: Showing running VMs with the `virsh list` command

With one command, we were able to list the VMs running on the server. In the example screenshot, you can see that I have a single VM running. If you'd also like to see non-running instances, simply add the `--all` option to the command.

We can manage the state of our VMs with any of the following commands:

- `virsh start vm-name`
- `virsh shutdown vm-name`
- `virsh suspend vm-name`
- `virsh resume vm-name`
- `virsh destroy vm-name`
- `virsh undefine vm-name`

The command syntax for `virsh` is extremely straightforward. By looking at the previous list of commands, you should be able to glean exactly what they do. The `virsh` commands allow us to do things such as `start`, `shutdown`, `suspend`, and `resume` a VM. The `virsh destroy` command is potentially destructive, as we'd use it when we want to halt a VM abruptly. It's essentially the same result as pulling a power cable from a physical server; it stops the instance immediately. You should only run that command when you are dealing with an unresponsive VM. Finally, the `virsh undefine` command deletes a VM, but you'll have to remove any associated disk files with the `rm` command. The default directory for disk files is `/var/lib/libvirt/images`, so you can look inside that directory for any disk files that belong to the VM you've deleted (they will be named the same as the VM).

That's not all `virsh` can do, however. We can actually create a VM with the `virsh` suite of commands as well. Learning how to do so is a good idea if you don't use Linux as your workstation operating system, or you don't have access to `virt-manager` for some reason. However, manually creating VM disk images and configuration is outside the scope of this chapter. The main goal is for you to familiarize yourself with managing VMs via `virsh`, and these simple basics will allow you to expand your knowledge further.

Summary

In this chapter, we took a look at virtualization, specifically with QEMU/KVM. We walked through the installation of KVM and the configuration required to get our virtualization server up and running. We walked through the process of creating a bridged network so that our VMs can be accessible from the rest of the network and created our first VM. In addition, although QEMU/KVM doesn't have its own solution for templating, we worked around that and created our own solution.

In *Chapter 17, Running Containers*, we'll take a look at containerization, which will include both Docker and LXD. Stay tuned!

Relevant video

- cloud-init guide (LearnLinuxTV): <https://linux.video/cloud-init>

Further reading

- Ubuntu `virsh` documentation: <https://learnlinux.link/u-virsh>
- *Mastering KVM Virtualization*, by Vedran Dakic et al. (Packt Publishing): <https://learnlinux.link/kvm-book>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



17

Running Containers

The IT industry never ceases to amaze me. Back when the concept of virtualization came about, it revolutionized the data center. Virtualization allowed us to run many small **Virtual Machines (VMs)** on one server, effectively allowing us to consolidate the equipment in our server racks. And just when we thought it couldn't get any better, the concept of containerization took the IT world by storm, allowing us to build portable instances of our software that not only improved how we deploy applications but also changed the way we develop them. In this chapter, we will cover the exciting world of containerization. This exploration will include:

- What is containerization?
- Understanding the differences between Docker and LXD
- Installing Docker
- Managing Docker containers
- Automating Docker image creation with Dockerfiles
- Managing LXD containers

To begin, let's explore what containerization is, how it differs from virtualization, and some considerations around how this technology might be implemented.

What is containerization?

In the last chapter, we covered virtualization. Virtualization allows us to run multiple *virtual* servers on one physical piece of hardware. We allocate CPU, RAM, and disk space to these VMs, and they run as if they were real servers. In fact, for all intents and purposes, a VM is a real server.

However, there are also weaknesses with VMs. Perhaps the most glaringly obvious is the fact that at least some of the resources you allocate to a VM are likely being wasted. For example, perhaps you've allocated 512 MB of RAM to a VM. What if the application only rarely uses more than 100 MB of RAM? That means most of the time, 412 MB of RAM that could otherwise be used for a useful purpose is just sitting idle. The same can be said of CPU usage. Nowadays, VM solutions do have ways of sharing unused resources, but effectively, resource efficiency is a natural weakness of the platform.

Containers, unlike VMs, are not actual servers. At least, not in the way you typically think about them (in terms of hardware). While VMs typically have one or more virtualized CPUs, containers share the CPU with the host. VMs also have their own kernel, but containers share the kernel of the host. Containers are still segregated, though. Just as a VM cannot access the host filesystem, a container can't either (unless you explicitly set it up to do so).

What is a container, then? It's probably best to think of a container as a filesystem rather than a VM. The container itself contains a file structure that matches that of the distribution it's based on. A container based on Ubuntu Server, for example, will have the same filesystem layout as a real Ubuntu Server installation on a VM or physical hardware. Imagine copying all the files and folders from an Ubuntu installation, putting them all in a single segregated directory, and having the binary contents of the filesystem executed as a program, without an actual operating system running.

To be fair, that description was an oversimplification of how containers actually run on an Ubuntu server, as the technology utilizes the functionality of the Linux kernel to isolate various components of a container from the rest of the system. However, a full discussion of those technologies is beyond the scope of this book. But understanding that such isolation exists within containers is something you should keep in mind, as keeping processes that are running within a container separate from other processes running on the host server is an important benefit.

Portability is another strength of **containerization**. With a container, you can literally pass it around to various members of your development team, and then push the container into production when everyone agrees that it's ready. The container itself will run exactly the same on each workstation, regardless of which operating system the workstation uses. To be fair, you can export and import VMs on any number of hosts, but containers make this process extremely easy. In fact, portability is at the core of the design of this technology.

The concept of containerization is not necessarily new. When Docker hit the scene, it took the IT world by storm, but it was by no means the first solution to offer containerization. LXC, and other technologies, predate it. It was, however, a clever marketing tactic with a cool-sounding brand that launched containerization into mainstream popularity. By no means am I saying that Docker is all hype, though. It's an awesome technology with many benefits. It's definitely worth using, and you may even find yourself preferring it to VMs.

The main difference with containerization is that each container generally does one thing. For example, perhaps a container holds a hosted website or contains a single application. VMs are often created to do many tasks, such as a web server that hosts ten websites. Containers, on the other hand, are generally used for one task each, though depending on the implementation you may see others going against this norm.

When should you use containers? I recommend you consider containers any time you're running a web app or some sort of service and you'd benefit from saving resources. The truth is, not all applications will run well in a container, but it's at least something to consider. Any time you're running an application that is typically accessed via a web browser, it's probably better off in a container rather than a VM. As an administrator, you'll most likely experiment with the different tools available to you and decide on the best tool for the job based on your findings.

Now that we understand the core concepts surrounding containers, let's explore the differences between two container technologies.

Understanding the differences between Docker and LXD

In this chapter, we're going to explore both Docker and LXD and see examples of containers running in both. Before we start working on that though, it's a good idea to understand some of the things that set each solution apart from the other.

Docker is probably the technology most of my readers have heard of. It seems as though you can't visit a single IT conference nowadays without it at least being mentioned. Docker is everywhere, and it runs on pretty much any platform. There's lots of documentation available for Docker and various resources you can utilize to deploy it. Docker utilizes a *layered* approach to containerization. Every change you make to the container creates a new layer, and these layers can form the base of other containers, thus saving disk space. More on that later.

LXD (pronounced Lex-D) finds its roots in **LXC**, so it's important to understand that first before we talk about LXD. LXC (pronounced Lex-C) is short for **Linux Containers** and is another implementation of containerization, similar to Docker. This technology, like similar solutions, uses the **control groups** (**cgroups**) feature of the Linux kernel, which isolates processes and is able to segregate them from one another. This enhances security, as processes should not be able to read data from other processes unless there's a good reason to.

LXC takes the concept of segregation even further, by creating an implementation of virtualization based solely on running applications in an isolated environment that matches the environment of an operating system. You can run LXC containers on just about every distribution of Linux available today.

LXD is also available for many Linux distributions, but it's treated as a first-class citizen in Ubuntu. This is because Canonical (the company behind Ubuntu) had a major hand in its development, and also offers commercial support for it. Since the software that makes LXD itself work is distributed via *snap* packages, that essentially means that any distribution of Linux that is able to install *snap* packages should be able to install LXD.

LXD takes LXC and gives it additional features that it otherwise wouldn't have, such as snapshots, ZFS support, and migration. LXD doesn't replace LXC; it actually utilizes it to provide its base technology. Perhaps the best way to think of LXD is as LXC with an additional management layer on top that adds additional features.

How does LXD/LXC differ from Docker? The main difference is that while they are both container solutions and solve the same goal in a very similar way, LXD is more similar to an actual VM while Docker tries harder to differentiate itself from that. In comparison, Docker containers are transactional (like I've already mentioned) and you generally have an `ENTRYPOINT` command that is run inside the container when you launch it. Essentially, LXD has a filesystem that you can directly access from the host operating system and has a simpler approach to containerization. You can think of LXC as a form of *machine* container that closely emulates a VM, and a Docker container is an *application* container that provides the foundation needed to run an application. Regardless of these differences, the technologies can be used in the same way and provide support for identical use cases.

When should you use Docker and when should you use LXD? I actually recommend you practice both since they're not overly difficult to learn. We will go over the basics of these technologies in this chapter. But to answer the question at hand, there are a few use cases where one technology may make more sense than the other. Docker is more of a general-purpose tool. You can run Docker containers on Linux, macOS, and even Windows. It's, therefore, a good choice if you want to create a container that runs everywhere. LXD is generally best for Linux environments, though Docker runs great in Linux too. The operating system you're running your container solution on is of little importance nowadays, since most people use a container service to run containers rather than an actual server that you manage yourself. In the future, if you get heavily into containerization, you may find yourself forgoing the operating system altogether and just running them in a service such as Amazon's **Elastic Container Service (ECS)**, which is one of a handful of cloud services that allow you to run containers without having to manage the underlying server.

Another benefit of Docker is **Docker Hub**, which you can use to download containers others have made or even upload your own for others to use. The benefit here is that if someone has already solved the goal you're trying to achieve, you can benefit from their work rather than starting from scratch, and they can also benefit from your work as well. This saves time and is often better than creating a solution by hand.

Always make sure to audit third-party resources before you put them to use in your organization. This includes (but isn't limited to) containers developed by a third party. You should understand how the container image was built, how secure the settings are, and whether or not there's anything built-in that might impose a security risk. Basically, some administrators will happily accept a container image as-is, but that practice can be very risky. Never deploy a container image that hasn't been audited for security.

Now that we understand not only the core concepts but the differences between the two containerization standards, let's take a look at Docker.

Installing Docker

Installing Docker is very fast and easy, so much so that it barely constitutes its own section. In the last chapter, we had to install several packages in order to get a **Kernel-based Virtual Machine (KVM)** virtualization server up and running as well as tweak some configuration files. In comparison, installing Docker is effortless, as you only need to install the `docker.io` package:

```
sudo apt install docker.io
```

Yes, that's all there is to it. Installing Docker was definitely much easier than setting up KVM, as we did in the previous chapter. Ubuntu includes Docker in its default repositories, so it's only a matter of installing this one package and its dependencies. You'll now have a new service installed on your machine, simply titled `docker`. In order to be useful, the service needs to be running. You can check to see whether or not it's already running with the following command:

```
systemctl status docker
```

Check the output of the previous command to see if the `docker` service is running. You should also check to see if the service is enabled. If not, you can start the service and also enable it at the same time with the following command:

```
systemctl enable --now docker
```

We also have the `docker` command available to us now, which allows us to manage our containers. By default, it does require root privileges, so you'll need to use `sudo` to use it. To make this easier, I recommend that you add your user account to the `docker` group before going any further. This will eliminate the need to use `sudo` every time you run a `docker` command. The following command will add your user account to the appropriate group:

```
sudo usermod -aG docker <yourusername>
```

After you log out and then log in again, you'll be able to manage Docker much more easily.

You can verify your group membership by simply running the `groups` command with no options, which should now show your user as a member of the `docker` group.

Well, that's it. Docker is installed, and your user account is a member of the `docker` group, so you're good to go. Wow, that was easy!

Now that we have Docker installed, let's start using it. It's a fun technology to learn, and in the next section, we'll explore a few examples.

Managing Docker containers

Now that Docker is installed and running, let's take it for a test drive. After installing Docker, we have the `docker` command available to use now, which has various sub-commands to perform different functions with containers. First, let's try out `docker search`:

```
docker search ubuntu
```

With Docker, containers are created from images. There are many pre-existing container images we can use, or we can build our own. The `docker search` command allows us to search for a container image that already exists and has been made available to us. Once we've chosen an image, we can download it locally and create container instances from it.

The ability of administrators to search for (and download) an existing container is just one of many great features Docker offers us. Although we can definitely build our own container images (and we will do so, right here in this chapter), sometimes it might make sense to use a pre-existing container image, rather than create a new one from scratch.

For example, you can install an NGINX container, simply named `nginx`. This is actually an official container image, so it should be trustworthy. We can tell that a container image is trustworthy by the verbiage **DOCKER OFFICIAL IMAGE** being present if you were to look up the image on the Docker Hub website at <https://hub.docker.com>. If we wanted to deploy a container running NGINX, doing so via the official image would save us a lot of time, especially compared to creating one from scratch. After all, why reinvent the wheel if you don't have to?

However, even if the container image comes from a trustworthy source, you should still audit it. With the NGINX example, we can be fairly confident that the image is safe and doesn't contain any unwanted objects, such as malware. However, there's no such thing as 100% trustworthy when it comes to security, so we should audit them anyway.

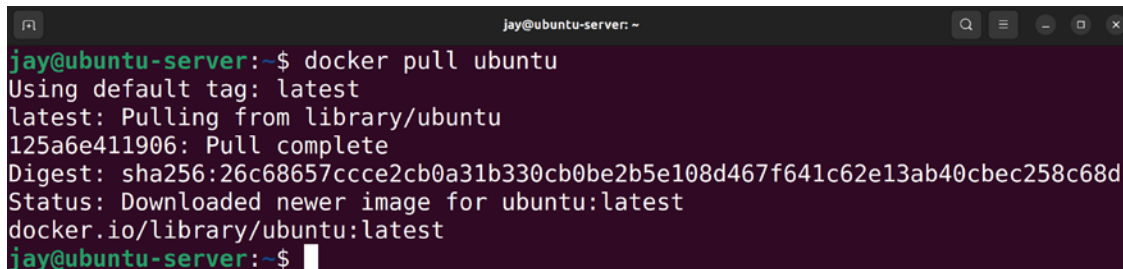
But how does this work? The `docker search` command will search Docker Hub, which is an online repository that hosts containers for others to download and utilize. You could search for containers based on other applications, or even other distributions such as Fedora or AlmaLinux, if you wanted to experiment. The command will return a list of Docker images available that meet your search criteria.

So what do we do with these images? An image in Docker is its closest equivalent to a VM or hardware image. It's a snapshot that contains the filesystem of a particular operating system or Linux distribution, along with some changes the author included to make it perform a specific task. This image can then be downloaded and customized to suit your purposes. You can choose to upload your customized image back to Docker Hub if you would like to contribute upstream. Every image you download will be stored on your machine so that you won't have to re-download it every time you wish to create a new container.

To pull down a Docker image for our use, we can use the `docker pull` command, along with one of the image names we saw in the output of our search command:

```
docker pull ubuntu
```

With the preceding command, we're pulling down the latest Ubuntu container image available on Docker Hub. The image will now be stored locally, and we'll be able to create new containers from it. The process will look similar to the following screenshot:

A terminal window titled 'jay@ubuntu-server: ~' showing the execution of 'docker pull ubuntu'. The output is as follows:

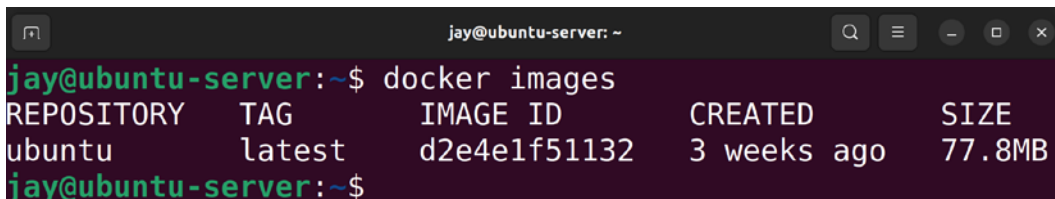
```
jay@ubuntu-server:~$ docker pull ubuntu
Using default tag: latest
latest: Pulling from library/ubuntu
125a6e411906: Pull complete
Digest: sha256:26c68657ccce2cb0a31b330cb0be2b5e108d467f641c62e13ab40cbec258c68d
Status: Downloaded newer image for ubuntu:latest
docker.io/library/ubuntu:latest
jay@ubuntu-server:~$
```

Figure 17.1: Downloading an Ubuntu container image

If you're curious as to which images you have saved locally, you can execute `docker images` to get a list of the Docker container images you have stored on your server:

```
docker images
```

The output will look similar to this:



```
jay@ubuntu-server: ~  
jay@ubuntu-server:~$ docker images  
REPOSITORY    TAG       IMAGE ID       CREATED        SIZE  
ubuntu        latest    d2e4e1f51132   3 weeks ago    77.8MB  
jay@ubuntu-server:~$
```

Figure 17.2: Listing installed Docker images

Notice the `IMAGE ID` in the output. If for some reason you want to remove an image, you can do so with the `docker rmi` command, and you'll need to use the ID as an argument to tell the command what to delete. The syntax would look similar to this if I was removing the image with the ID shown in the screenshot:

```
docker rmi d2e4e1f51132
```

Once you have a container image downloaded to your server, you can create a new container from it by running the `docker run` command, followed by the name of your image and an application within the image to run. An application run from within a Docker container is known as an `ENTRYPOINT`, which is just a fancy term to describe an application a particular container is configured to run. You're not limited to the `ENTRYPOINT` though, and not all containers actually have an `ENTRYPOINT`. You can use any command in the container that you would normally be able to run in that distribution. In the case of the Ubuntu container image we downloaded earlier, we can run `bash` with the following command so that we can get a prompt and enter any command(s) we wish:

```
docker run -it ubuntu /bin/bash
```

Once you run that command, you're now interacting with a shell prompt from within your container. From here, you can run commands you would normally run within a real Ubuntu machine, such as installing new packages, changing configuration files, and more. Go ahead and play around with the container, and then we'll continue with a bit more theory on how this is actually working.

There are some potentially confusing aspects of Docker we should get out of the way first before we continue with additional examples. The thing that's most likely to confuse newcomers to Docker is how containers are created and destroyed. When you execute the `docker run` command against an image you've downloaded, you're actually creating a container. Therefore, the image you downloaded with the `docker pull` command wasn't an actual container itself, but it becomes a container when you run an instance of it. When the command that's being run inside the container finishes, the container goes away. Therefore, if you were to run `/bin/bash` in a container and install a bunch of packages, those packages would be wiped out as soon as you exited the container.

You can think of a Docker image as a “blueprint” for a container that can be used to create running containers. Every container you run has a container ID that differentiates it from others. If you want to remove a persistent container, for example, you would need to reference this ID with the `docker rm` command. This is very similar to the `docker rmi` command that’s used to remove container images.

To see the container ID for yourself, you’ll first need to exit the container if you’re currently running one. There are two ways of doing so. First, you could press `Ctrl + d` to disconnect, or even type `exit` and press `Enter`. When you exit the container, you’re removing it (Docker containers only typically exist while running). When you run the `docker ps` command (which is the command you’ll use any time you want a list of containers on your system), you won’t see it listed. Instead, you can add the `-a` option to see all containers listed, even those that have been stopped.

You’re probably wondering, then, how to exit a container and not have it go away. To do so, while you’re attached to a container, press `Ctrl + p` and then press `q` (don’t let go of the `Ctrl` key while you press these two letters). This will drop you out of the container, and when you run the `docker ps` command (even without the `-a` option), you’ll see that it’s still running.

The `docker ps` command deserves some attention. The output will give you some very useful information about the containers on your server, including the `CONTAINER ID` that was mentioned earlier. In addition, the output will contain the `IMAGE` it was created from, the `COMMAND` being run when the container was `CREATED`, and its `STATUS`, as well as any `PORTS` you may have forwarded. The output will also display randomly generated names for each container, which are usually quite comical. As I was going through the process of creating containers while writing this section, the code names for my containers were `tender_cori`, `serene_mcnulty`, and `high_goldwasser`. This is just one of the many quirks of Docker, and some of these can be quite humorous.

The important output of the `docker ps -a` command is the `CONTAINER ID`, the `COMMAND`, and the `STATUS`. The ID, which we already discussed, allows you to reference a specific container to enable you to run commands against it. `COMMAND` lets you know what command was being run. In our example, we executed `/bin/bash` when we started our containers.

If we have any containers that were stopped, we can resume a container with the `docker start` command, giving it a container ID as an argument. Your command will end up looking similar to this:

```
docker start d2e4e1f51132
```

The output will simply return the ID of the container, and then drop you back to your shell prompt—not the shell prompt of your container, but that of your server. You might be wondering at this point, how do I get back to the shell prompt for the container? We can use `docker attach` for that:

```
docker attach d2e4e1f51132
```

The `docker attach` command is useful because it allows you to attach your shell to a container that is already running. Most of the time, containers are started automatically instead of starting with `/bin/bash` as we have done. If something were to go wrong, we may want to use something like `docker attach` to browse through the running container to look for error messages. It’s very useful.

Speaking of useful, another great command is `docker info`. This command will give you information about your implementation of Docker, such as letting you know how many containers you have on your system, which should be the number of times you've run the `docker run` command unless you cleaned up the previously run containers with `docker rm`. Feel free to take a look at its output and see what you can learn from it.

Getting deeper into the subject of containers, it's important to understand what a Docker container is and what it isn't. A container is not a service running in the background, at least not inherently. A container is a collection of namespaces, such as a namespace for its filesystem or users. As discussed earlier in this chapter, containers are isolated from the rest of the server by utilizing technology within the Linux kernel. When you disconnect without a process running within the container, there's no reason for it to run, since its namespace is empty. Thus, it stops. If you'd like to run a container in a way that is similar to a service (it keeps running in the background), you would want to run the container in **detached mode**. Basically, this is a way of telling your container to run this process and to not stop running it until you tell it to. Here's an example of creating a container and running it in detached mode:

```
docker run -dit ubuntu /bin/bash
```

After running the previous command, Docker will print a container ID, and then drop back to your command prompt. You can then see that the container is running with the `docker ps` command, so use `docker attach` along with the container ID to connect to it and run commands.

Normally, we use the `-it` options to create a container. This is what we used a few examples ago. The `-i` option triggers interactive mode, while the `-t` option gives us a pseudo-TTY. At the end of the command, we tell the container to run the Bash shell. The `-d` option runs the container in the background.

It may seem relatively useless to have another Bash shell running in the background that isn't actually performing a task. But these are just simple examples to help you get the hang of Docker. A more common use case may be to run a specific application. In fact, you can even serve a website from a Docker container by installing and configuring Apache within the container, including a virtual host. The question then becomes: how do you access the container's instance of Apache within a web browser? The answer is **port redirection**, which Docker also supports. Let's give this a try.

First, let's create a new container in detached mode. Let's also redirect port 80 within the container to port 8080 on the host:

```
docker run -dit -p 8080:80 ubuntu /bin/bash
```

The command will output a container ID. This ID will be much longer than you're accustomed to seeing. This is because when we run `docker ps -a`, it only shows shortened container IDs. You don't need to use the entire container ID when you attach; you can simply use part of it as long as it's long enough to be different from other IDs:

```
docker attach dfb3e
```

Here, I've attached to a container with an ID that begins with `dfb3e`. This will connect my shell to a Bash shell within the container.

Let's install Apache. We've done this before, but there are a few differences that you'll see. First, if you simply run the following command to install the `apache2` package as we would normally do, it may fail for one or two reasons:

```
sudo apt install apache2
```

The two problems here are first that `sudo` isn't included by default in the Ubuntu container, so it won't even recognize the `sudo` part of the command. When you run `docker attach`, you're actually attaching to the container as the `root` user, so the lack of `sudo` won't be an issue anyway. Second, the repository index in the container may be out of date, if it's even present at all. This means that `apt` within the container won't even find the `apache2` package. To solve this, we'll first update the repository index:

```
apt update
```

Then, install `apache2` using the following command:

```
apt install apache2
```

You may be asked to set your time zone or geographic location during the installation of packages. If so, go ahead and enter each prompt accordingly.

Now we have Apache installed in our container. We don't need to worry about configuring the default sample web page or making it look nice. We just want to verify that it works. Let's start the service:

```
/etc/init.d/apache2 start
```

After running that command, Apache should be running within the container.

The previous command is definitely not our normal way of starting services. Typically, we'd use a command like `systemctl start apache2`, but there's no actual `init` system inside a container, so running `systemctl` commands will not work as they normally would. Always refer to any documentation that may exist for a container you're attempting to run, regarding how to start an application it may contain.

Apache should be running within the container. Now, press `Ctrl + p` and then press `q` (don't let go of the `Ctrl` key while you press these two letters) to exit the container, but allow it to keep running in the background. You should be able to visit the sample Apache web page for the container by navigating to `localhost:8080` in your web browser. You should see the default `It works!` page of Apache.

Congratulations, you're officially running an application within a container:

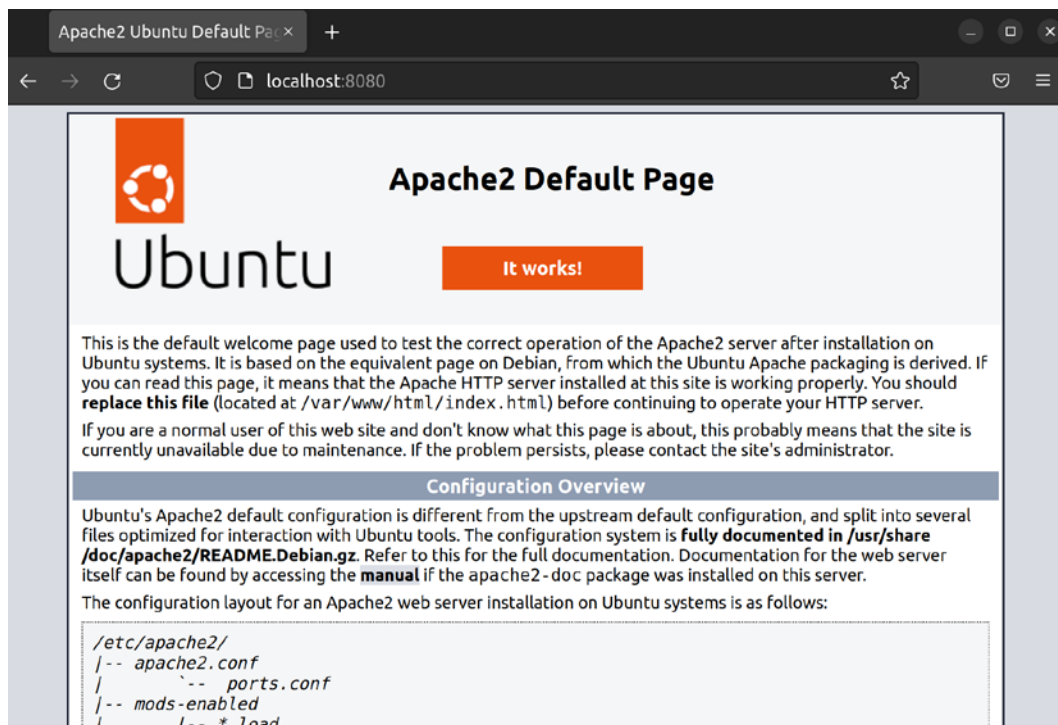


Figure 17.3: The default Apache start page, running from within a container

As your Docker knowledge grows, you'll want to look deeper into the concept of an ENTRYPOINT. An ENTRYPOINT is a preferred way of starting applications in a Docker container. In our examples so far, we've used an ENTRYPOINT of `/bin/bash`. While that's perfectly valid, an ENTRYPOINT is generally a Bash script that is configured to run the desired application and is launched by the container.

Our Apache container is running happily in the background, responding to HTTP requests over port 8080 on the host. But what should we do with it at this point? We can create our own image from it so that we can simplify deploying it later. To be fair, we've only installed Apache inside the container, so it's not saving us that much work. In a real production environment, you may have a container running that needed quite a few commands to set it up. With an image, we can have all of that work baked into the image, so we won't have to run any setup commands we may have each time we want to create a container. To create a container image, let's grab the container ID of a running container by running the `docker ps` command. Once we have that, we can now create a new image of the container with the `docker commit` command:

```
docker commit <Container ID> ubuntu/apache-server:1.0
```

That command will return us the ID of our new image. To view all the Docker images available on our machine, we can run the `docker images` command to have Docker return a list. You should see the original Ubuntu image we downloaded, along with the one we just created. We'll first see a column for the repository the image came from; in our case, it is Ubuntu. Next, we see the tag. Our original Ubuntu image (the one we used `docker pull` to download) has a tag of `latest`. We didn't specify that when we first downloaded it; it just defaulted to `latest`. In addition, we see an image ID for both, as well as the size.

To create a new container from our new image, we just need to use `docker run`, but specify the tag and name of our new image. Note that we may already have a container listening on port `8080`, so this command may fail if that container hasn't been stopped:

```
docker run -dit -p 8080:80 ubuntu/apache-server:1.0 /bin/bash
```

Speaking of stopping a container, I should probably show you how to do that as well. As you can probably guess, the command is `docker stop` followed by a container ID:

```
docker stop <Container ID>
```

This will send the `SIGTERM` signal to the container, followed by `SIGKILL` if it doesn't stop on its own after a delay.

Admittedly, the Apache container example was fairly simplistic, but it does the job as far as showing you a working example of a container that is actually somewhat useful. Before continuing on, think for a moment of all the use cases you can use Docker for in your organization. It may seem like a very simple concept (and it is), but it allows you to do some very powerful things. Perhaps you'll want to try to containerize your organization's intranet page or some sort of application. The concept of Docker sure is simple, but it can go a long way with the right imagination.

Before I close out this section, I'll give you a personal example of how I implemented a container at a previous job. At this organization, I worked with some Embedded Linux software engineers who each had their own personal favorite Linux distribution. Some preferred Ubuntu, others preferred Debian, and a few even ran Gentoo. This in and of itself wasn't necessarily an issue—sometimes it's fun to try out other distributions. But for developers, a platform change can introduce inconsistency, and that's not good for a software project. The included build tools are different in each distribution of Linux because they all ship different versions of all the development packages and libraries. The application this particular organization developed was only known to compile properly in Debian, and newer versions of the compiler posed a problem for the application. My solution was to provide each developer with a Docker container based on Debian, with all the build tools that they needed to perform their job baked in. At this point, it no longer mattered which distribution they ran on their workstations. The container was the same no matter what they were running. Regardless of what their underlying operating system was, they all had the same tools. This gave each developer the freedom to run their preferred distribution of Linux (or even macOS), and it didn't impact their ability to do their job. I'm sure there are some clever use cases you can come up with for implementing containerization.

Now that we understand the basics of Docker, let's take a look at automating the process of building containers.

Automating Docker image creation with Dockerfiles

I've mentioned previously in this book that anything worth having a server do more than once should be automated, and building a Docker container is no exception. A Dockerfile is a neat way of automating the building of Docker images by creating a text file with a set of instructions for their creation. Docker is able to take this file, execute the commands it contains, and build a container. It's magic.

The easiest way to set up a Dockerfile is to create a directory, preferably with a descriptive name for the image you'd like to create (you can name it whatever you wish, though), and inside it create a text file named Dockerfile. For a quick example, copy this text into your Dockerfile and I'll explain how it works:

```
FROM ubuntu
MAINTAINER Jay <jay@somewhere.net>
# Avoid confirmation messages
ARG DEBIAN_FRONTEND=noninteractive
# Update the container's packages
RUN apt update; apt dist-upgrade -y
# Install apache2 and vim
RUN apt install -y apache2 vim-nox
# Start Apache
ENTRYPOINT apache2ctl -D FOREGROUND
```

Let's go through this Dockerfile line by line to get a better understanding of what it's doing:

```
FROM ubuntu
```

We need an image to base our new image on, so we're using Ubuntu as a starting point.

This will cause Docker to download the `ubuntu:latest` image from Docker Hub, if we haven't already downloaded it locally. If we do have it locally, it will just use the locally cached version.

```
MAINTAINER Jay <myemail@somewhere.net>
```

Here, we're setting the maintainer of the image. Basically, we're declaring its author. This is optional, so you don't need to include that if you don't want to.

```
# Avoid confirmation messages
```

Lines beginning with a hash symbol (#) are ignored, so we are able to create comments within the Dockerfile. This is recommended to give others a good idea of what your Dockerfile is doing.

```
ARG DEBIAN_FRONTEND=noninteractive
```

Here, we're setting an environment variable that, in this case, sets the environment to `noninteractive`. The reason we do this is that the process of installing a package while building a Docker container should be automatic; if a prompt comes up while a package is being installed and asks you a question, your input will not pass through and the process will hang.

With this environment variable, we're clarifying that we want to be in noninteractive mode so that the default answers to any questions that come up will be used and we won't be prompted.

```
RUN apt update; apt dist-upgrade -y
```

With the `RUN` command, we're telling Docker to run a specific command while the image is being created. In this case, we're updating the image's repository index and performing a full package update to ensure the resulting image is as fresh as can be. The `-y` option is provided to suppress any requests for confirmation while installing the packages. Despite the fact that we set `noninteractive` mode earlier, `apt` will still try to confirm changes interactively, and the `-y` option suppresses that.

```
RUN apt install -y apache2 vim-nox
```

Next, we're installing both `apache2` and `vim-nox`. The `vim-nox` package isn't required, but I personally like to make sure all of my servers and containers have it installed. I mainly included it here to show you that you can install multiple packages in one line.

```
ENTRYPOINT apache2ctl -D FOREGROUND
```

I mentioned the concept of an `ENTRYPOINT` earlier, which again is where we clarify which application should run when the container starts. The `apache2ctl` command is a wrapper command for Apache that allows administrators to control the finer points of running the Apache daemon. A full walk-through of this command is beyond the scope of this chapter, but we're using it here because we want Apache to automatically start with the container, and `apache2ctl` is one method of doing that without relying on `systemctl` (which the container doesn't have).

Great, now we have a `Dockerfile`. So what do we do with it? Well, turn it into an image of course! To do so, we can use the `docker build` command, which can be executed from within the directory that contains the `Dockerfile`. Here's an example of using the `docker build` command to create an image tagged `packt/apache-server:1.0`:

```
docker build -t learnlinux/v/apache-server:1.0 .
```

Once you run that command, you'll see Docker create the image for you, running each of the commands you asked it to. The image will be set up just the way you like. Basically, we just automated the entire creation of the Apache container we used as an example in this section. If anything goes wrong, Docker will print an error to your shell. You can then fix the error in your `Dockerfile` and run it again, and it will continue where it left off.

Once complete, we can create a container from our new image:

```
docker run -dit -p 8080:80 learnlinux/v/apache-server:1.0
```

Almost immediately after running the container, the sample Apache site will be available on `localhost:8080` on the host. With a `Dockerfile`, you'll be able to automate the creation of your Docker images. That was easy, wasn't it? There's much more you can do with `Dockerfiles`; feel free to peruse Docker's official documentation to learn more. Exploration is key, so give it a try and experiment with it.

Managing LXD containers

With Docker out of the way, let's take a look at how to run containers with LXD. Let's dive right in and install the required package:

```
sudo snap install lxd
```

As you can see, installing LXD is just as easy as installing Docker. In fact, managing containers with LXD is very straightforward as well, as you'll soon see. Installing LXD gives us the `lxc` command, which is the command we'll use to manage LXD containers. Before we get going though, we should add our user account to the `lxd` group:

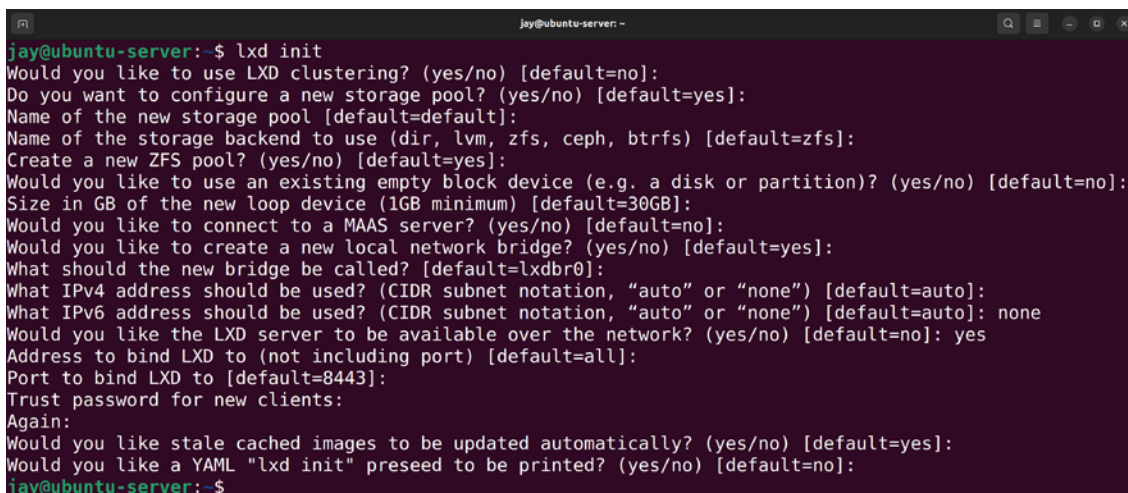
```
sudo usermod -aG lxd <yourusername>
```

Make sure you log out and log in for the changes to take effect. Just like with the `docker` group with Docker, the `lxd` group will allow our user account to manage LXD containers.

Next, we need to initialize our new LXD installation. We'll do that with the `lxd init` command:

```
lxd init
```

The process will look similar to the following screenshot:



```
jay@ubuntu-server:~$ lxd init
Would you like to use LXD clustering? (yes/no) [default=no]:
Do you want to configure a new storage pool? (yes/no) [default=yes]:
Name of the new storage pool [default=default]:
Name of the storage backend to use (dir, lvm, zfs, ceph, btrfs) [default=zfs]:
Create a new ZFS pool? (yes/no) [default=yes]:
Would you like to use an existing empty block device (e.g. a disk or partition)? (yes/no) [default=no]:
Size in GB of the new loop device (1GB minimum) [default=30GB]:
Would you like to connect to a MAAS server? (yes/no) [default=no]:
Would you like to create a new local network bridge? (yes/no) [default=yes]:
What should the new bridge be called? [default=lxdbr0]:
What IPv4 address should be used? (CIDR subnet notation, "auto" or "none") [default=auto]:
What IPv6 address should be used? (CIDR subnet notation, "auto" or "none") [default=auto]: none
Would you like the LXD server to be available over the network? (yes/no) [default=no]: yes
Address to bind LXD to (not including port) [default=all]:
Port to bind LXD to [default=8443]:
Trust password for new clients:
Again:
Would you like stale cached images to be updated automatically? (yes/no) [default=yes]:
Would you like a YAML "lxd init" preseed to be printed? (yes/no) [default=no]:
jay@ubuntu-server:~$
```

Figure 17.4: Setting up LXD with the `lxd init` command

The `lxd init` command will ask us a series of questions regarding how we'd like to set up LXD. The defaults are mostly fine for everything, and for the size of the pool, I just used the default of 30 GB, but you can use whatever size you want to. I set `ipv6` to `none` during the setup since my network doesn't utilize that, and I also decided to make `lxd` available over the network.

Even though we chose the defaults for most of the questions, they'll give you a general consensus of some of the different options that LXD makes available for us. For example, we can see that LXD supports the concept of a storage pool, which is one of its neater features.

Here, we're creating a default storage pool with a filesystem format of `zfs`, which is a filesystem that is used on actual hard disks. During the setup process, LXD sets up the storage pool, network bridge, IP address scheme, and basically everything we need to get started.

Now that LXD is installed and set up, we can configure our first container:

```
lxc launch ubuntu:22.04 mycontainer
```

With that simple command, LXD will now download the root filesystem for this container and set it up for us. Once done, we'll have an LXD container based on Ubuntu 22.04 running and available for use. This is different than Docker, which only sets up an image by default, making us run it manually. During this process, we gave the container the name of `mycontainer`. The process should be fairly easy to follow so far.

You might be wondering why we used a `lxc` command to create a container since we're learning about LXD here. As I mentioned earlier, LXD is an improvement over LXC, and as such, it uses `lxc` commands for management. Commands that are specific to the LXD layer will be `lxd`, and anything specific to container management will be done with `lxc`.

When it comes to managing containers, there are several types of operations you will want to perform, such as listing containers, starting a container, stopping a container, deleting a container, and so on. The `lxc` command suite is very easy and straightforward. Here is a table listing some of the most common commands you can use, and I'm sure you'll agree that the command syntax is very logical. For each example, you substitute `<container>` with the name of the container you created:

Goal	Command
List the containers	<code>lxc list</code>
Start a container	<code>lxc start <container></code>
Stop a container	<code>lxc stop <container></code>
Remove a container	<code>lxc delete <container></code>
List the downloaded images	<code>lxc image list</code>
Remove an image	<code>lxc image delete <image_name></code>

With all the basics out of the way, let's jump into our container and play around with it. To open a shell to the container we just created, we would run the following:

```
lxc exec mycontainer bash
```

In the preceding command, `exec` tells the container we want to execute a command, `mycontainer` is the name of the container that we want to execute something against, and the specific command we want to execute is `bash`. After you execute that command, it immediately runs `bash` from the container as `root`. From here, you can configure the container as you need to by installing packages, setting up services, or whatever else you may need to do in order to make the container conform to the purpose you have for it. In fact, the process of customizing the container for redeployment is actually easier than it is with Docker.

Unlike with Docker, changes are not wiped when you exit a container, and you don't have to exit it in a particular way to avoid losing your changes. We also don't have layers to deal with in LXD, which you may or may not be happy about (layers in Docker containers can make deployments faster, but when previously run containers aren't cleaned up, the number of layers can look messy).

The Ubuntu image we used to create our container includes a default user account, `ubuntu`. This is similar to some VPS providers, which also include an `ubuntu` user account by default (Amazon EC2 is an example of this). If you prefer to log in as this user rather than `root`, you can do that with this command:

```
lxc exec mycontainer -- su --login ubuntu
```

The `ubuntu` user has access to `sudo`, so you'll be able to run privileged tasks with no issue.

To exit the container, you can press `Ctrl + d` on your keyboard, or simply type `exit`. Feel free to log in to the container and make some changes and experiment. Once you have the container set up the way you like it, you may want the container to automatically start up when you boot your server. This is actually very easy to do:

```
lxc config set mycontainer boot.autostart 1
```

With the preceding command, we're setting `boot.autostart` to `1`, which turns on that particular feature. Similar to a Boolean variable for those that are familiar with programming, `1` means "on" and `0` means "off." After setting this config value, your newly created container will now start up with the server anytime it's booted.

Now, let's have a bit of fun. Feel free to install the `apache2` package in your container. Similar to Docker, I've found that you will probably want to run `apt update` to update your package listings first, as I've seen failures installing packages on a fresh container solely because the indexes were stale. So just run this to be safe:

```
sudo apt update && sudo apt install apache2
```

Now, you should have Apache installed and running in the container. Next, we need to grab the IP address of the container. Yes, you read that right, LXD has its own IP address space for its containers, which is very neat. Simply run `ip addr show` (the same command you'd run in a normal server), and it will display the IP address information. On the same machine that's running the container, you can visit this IP address to see the default Apache web page. If you're running the container on a server with no graphical user interface, you can use the `curl` command to verify that it's working:

```
curl <container_ip_address>
```

Although we have Apache running in our container, we can see that it's not very useful yet. The web page is only available from the machine that's hosting the container. This doesn't help us much if we want users in our local network or even from the outside internet to be able to reach our site. We could set up firewall rules to route traffic to it, but there's an easier way—creating a profile for external access.

I mentioned earlier that even though LXD is a containerization technology, it shares some of its feature set with VMs, basically giving you VM-like features in a non-VM environment. With LXD, we can create a profile to allow it to get an IP address from your DHCP server and route traffic directly through your LAN, just as with a physical device that you connect to your network.

Before continuing, you'll need a bridge connection set up on your server. This is done in software via Netplan and was discussed as part of the previous chapter. If you list your network interfaces (`ip addr show`), you should see a `br0` connection. If you don't have this configured, refer back to *Chapter 16, Virtualization*, and refer to the *Bridging the VM network* section there. Once you've created this connection, you can continue on.

Some network cards do not support bridging, especially with some Wi-Fi cards. If you're unable to create a bridge on your hardware, the following section may not work for you. Consult the documentation for your hardware to ensure your network card supports bridging.

To create the profile we'll need in order to enable external access to our containers, we'll use the following command:

```
lxc profile create external
```

We should see output similar to the following:

```
Profile external created
```

Next, we'll need to edit the profile we just created. The following command will open the profile in a text editor so that you can edit it:

```
lxc profile edit external
```

Inside the profile, we'll replace its text with this:

```
description: External access profile
devices:
  eth0:
    name: eth0
    nictype: bridged
    parent: br0
    type: nic
```

From this point forward, we can launch new containers with this profile with the following command:

```
lxc launch ubuntu:22.04 mynewcontainer -p default -p external
```

Notice how we're applying two profiles, `default` and then `external`. We do this so that the values in `default` can be loaded first, followed by the second profile so that it overrides any conflicting parameters that may be present.

We already have a container, though, so you may be curious how we can edit the existing one to take advantage of our new profile. That's simple:

```
lxc profile add mycontainer external
```

From this point forward, assuming the host bridge on your server has been configured properly, the container should be accessible via your local LAN. You should be able to host a resource, such as a website, and have others be able to access it. This resource could be a local intranet site or even an internet-facing website.

As far as getting started with LXD is concerned, that's essentially it. LXD is very simple to use, and its command structure is very logical and easy to understand. With just a few simple commands, we can create a container, and even make it externally accessible. Canonical has many examples and tutorials available online to help you push your knowledge even further, but with what you've learned so far, you should have enough practical knowledge to roll out this solution in your organization.

Summary

Containers are a wonderful method of hosting applications. You can spin up more containers on your hardware than you'd be able to with VMs, which will definitely save resources. While not all applications can be run inside containers, it's a very useful tool to have available. In this chapter, we looked at both Docker and LXD. While Docker is better for cross-platform applications, LXD is simpler to use but is very flexible. We started out by discussing the differences between these two solutions, then we experimented with both creating containers and looking at how to manage them.

In the next chapter, we will expand our knowledge of containers even further and take a look at orchestration, which allows us to manage multiple containers more efficiently. This will be the chapter where all of the concepts relating to containers come together.

Relevant videos

- Docker Essentials tutorial series (LearnLinuxTV): <https://linux.video/docker-essentials>
- Getting started with LXD (LearnLinuxTV): <https://linux.video/lxd-guide>

Further reading

- Docker documentation page: <https://learnlinux.link/d-docs>
- LXD introduction: <https://learnlinux.link/lxd-intro>
- LXD documentation: <https://learnlinux.link/c-lxd>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



18

Container Orchestration

In the previous chapter, we started learning concepts around containerization. We've learned what containers are and how they differ from **Virtual Machines (VMs)**, as well as how to run two different types of containers (Docker and LXD). As you are now aware, containers are typically lightweight (which means you run a larger number of them than VMs on the same hardware) and are easy to manage with a command syntax that's rather logical, such as `docker run myapp` to launch a container named `myapp`. Depending on the size of your organization, you may only need to run one or two containers to suit your needs, or perhaps you plan on scaling up to hundreds of them. While it's rather simple to maintain a small number of containers, the footprint can quickly expand and become much harder to keep track of.

In this chapter, we'll start looking into the concept of **Container Orchestration**, which can help us to better maintain the containers that we run. Without such orchestration, the onus is on you, the administrator, to ensure your containers are running properly. While someone still needs to be responsible for mission-critical containers, orchestration provides us with tools we can use to manage them much more efficiently. In addition, orchestration allows us to create an entire cluster that's easily scalable, so we're better equipped to handle the demand of our users or clients. As we navigate this topic, we will work on:

- Container orchestration
- Preparing a lab environment for Kubernetes testing
- Utilizing MicroK8s
- Setting up a Kubernetes cluster
- Deploying containers via Kubernetes

Excited? I know I am—containerization can be a lot of fun to learn and work with. We'll even create our very own cluster in this chapter! But before we can do that, let's make sure we have an understanding of what container orchestration is.

Container orchestration

In the last chapter, we covered the basics of running containers on your server. Of special note is the coverage of Docker, which will play a very important role in this chapter. We saw how to pull a Docker image as well as how to use such an image to create a container. There are many more advanced concepts we can learn when it comes to Docker, but understanding the essentials is good enough for the scope of this chapter. And now that we know how to run containers, looking further into how to more efficiently manage them is a logical next step.

Traditionally, as an administrator, you'll ensure the critical apps and services for your organization are always healthy and available. If a critical resource stops working for any reason, it falls on you to return it to a healthy state. Regardless of whether we're utilizing applications on a physical server, or in a VM or container, this need doesn't change—production apps need to be available at all times with minimal or no downtime. Specific to containers, what orchestration does for us is help us maintain them much more efficiently. Orchestration allows us to not only manage our containers in one place, but it also provides us with additional tooling we can use to more intelligently handle load and recover from faults.

Consider this example: let's assume you work at an organization that has an important website that needs to be available online at all times, and it's currently set up inside a VM. As the administrator, you test the application by running it inside a Docker container and discover that it not only functions the same as in the VM, it also requires less of the server's memory to run it in a container, and it's even more responsive than before. Everyone at your company is impressed, and the project of converting your company's site to run inside a container is a complete success.

Now, let's assume your organization is getting ready to release a brand new version of your company's product. It's expected that the demand for your website will increase ten times for at least the first few weeks after the debut. To address this, you can launch however many additional containers you feel will handle the expected load, and then set up a load balancer to spread traffic evenly between them. When the excitement over the new release winds down, you can remove the newly added containers to return to your original population. We haven't gone over load balancers yet, but these are useful for spreading traffic between multiple nodes. This feature is built into Kubernetes, so you won't need to install anything extra to take advantage of this.

If the launch of your organization's newly updated product is expected to go live at 1:00am, you'd have to make sure you're awake then and execute the necessary commands to launch the extra containers and deploy the load balancer. You would then watch the containers for a while and ensure they're stable. Perhaps you've already tested this in a testing environment, so you have reasonable confidence that the maintenance will go smoothly. After you successfully launch the containers, you set a reminder for yourself in your calendar to log back in after two weeks to undo the changes.

While that approach can technically work and result in a successful rollout, it's not very efficient. Sure, you *can* log in at 1:00am to launch the extra containers and deploy the load balancer, but is that an effective use of your time? What if you're very sleepy at that time and make a mistake that results in a failure during an important launch? And when the launch window is over and the load returns to normal, you may or may not see the alert that you've intended to use to remind yourself and lower the container count.

In that situation, you could end up with an expensive bill since your server would be using extra energy to run containers that were no longer needed. Even worse, manually managing your containers means that it can't handle a situation where the load increases unexpectedly. Despite our best intentions, any process that is run manually may or may not work out well; we're only human after all.

With container orchestration, we essentially delegate the process of running containers to an application that will automatically create additional containers as demand increases and remove unneeded containers when demand winds down. Orchestration also simplifies the process of setting up applications, by giving us tools we can use to ensure a particular number of containers are running at a minimum and we can set a maximum for when load and demand spikes. This empowers us to have infrastructure that grows and shrinks automatically to match the needs of our users. In addition, container orchestration allows us to automatically heal from failures. If a container runs into a problem and fails for some reason, it will be deleted and recreated from the image.

As you can see, orchestration gives us additional value, it can actually save you time and effort. To be fair, some of the features mentioned are part of containerization itself and not necessarily specific to orchestration, but the latter does give us the ability to manage these tools much more efficiently. Is container orchestration for everyone? No technology meets 100% of all use-cases, but it's certainly something to consider. If you only run a single container and have no plans to run another, then a technology such as orchestration is overkill—it would take you more time to manage the cluster than it would to manage the container itself. Use your best judgment.

If you do decide to implement containerization, **Kubernetes** (often abbreviated **K8s**) is a very popular solution for container orchestration and is what we'll be exploring in this chapter. What Kubernetes allows us to do is create deployments for our applications, providing us with fine-tuned control we can leverage to determine the actual behavior of our applications. It can automatically re-spawn a container if it stops working for any reason, ensure the number of containers running meets the demand, and spread our workloads across many worker nodes so that no one server becomes overwhelmed with running too many containers. Kubernetes is very popular and is not specific to Ubuntu Server. However, utilizing a technology such as Kubernetes in Ubuntu is one of many ways that we can use to extend the platform.

What do you need in order to set up a Kubernetes cluster? That's likely a logical question to have at this point. In the next section, we'll cover some of the considerations to make before deciding how to set it up.

Preparing a lab environment for Kubernetes testing

In an organization, planning a roll-out of an entire Kubernetes cluster can be fairly involved—you may have to purchase hardware and also analyze your existing environment and understand how containerization will fit in. It's possible that some applications you want to run aren't a good fit for containers; some don't support running in a container at all. Assuming you've already checked the documentation for the applications you are wanting to run in containers and came to the conclusion that such a technology is supported, the next step is procuring the hardware (if you don't already have a place to run it) and then setting up the cluster.

Specific to us in this book, we don't need to contact a server vendor and submit a purchase order to simply test out the technology. If you are actually involved with the rollout of container orchestration at your organization, then it's a fun project to work on. But for the purposes of this book, let's discuss some of the details around what's needed to set up a personal lab in order to set up Kubernetes and start learning the platform.

The best rule of thumb when setting up a lab for testing software is to try to use what you have available. To create a Kubernetes cluster, we'll need one Ubuntu machine to act as the controller, and one or more additional servers to be used as worker nodes. As you'll learn later in the book, the goal of Kubernetes is to schedule containers to run in **Pods** on worker nodes. Pods in Kubernetes are a collection of one or more containers, and every container is run in a pod. With more than one worker node, you can benefit from being able to run more pods (and as an extension, run more containers) and having load spread between multiple workers to have the most control over how your applications are served to the end-user.

But on what type of device should you run Kubernetes on in order to test it out and learn it? For this, we have the same possibilities as we discussed when installing Ubuntu Server back in *Chapter 1, Deploying Ubuntu Server*—we can use VMs, physical servers, spare desktops or laptops, as well as Raspberry Pi's (or any combination of those). Again, use whatever you have available. For a rollout in an organization, you may end up using a virtualization server for the controller and worker nodes, or perhaps physical servers. One of my favorite platforms for Kubernetes is the Raspberry Pi, believe it or not. I've been running a Kubernetes cluster in production for several years that consists of only Raspberry Pi 4 units with complete success. If nothing else, purchasing a few Pi's is a relatively low barrier to entry. You can even utilize a cloud provider if you'd like, though doing so goes beyond the scope of this chapter as cloud providers generally have their own tools in place to manage clusters.

In general, it's recommended to have a controller node and a handful of workers. A single worker will suffice, but in this chapter, I'll show the process of setting up two workers to better understand how the platform scales. On your end, you can use a single controller and worker, or set up however many workers you'd like. One of the best things about Kubernetes is that you aren't stuck with the number of workers you set up initially, you can add more nodes to your cluster as time goes on. An organization may start with just a few workers but add additional ones as the need arises.

As far as resources go, requirements for Kubernetes are relatively modest. The node that's designated as the controller should have a minimum of two CPUs or one CPU with at least two cores. Obviously, it's better if you have more than that, such as a quad-core CPU for the controller, but two cores are the minimum. The worker nodes can have a single CPU core each, but the more cores they have, the more Pods they'll be able to run. When it comes to RAM, I recommend a minimum of 2 GB of memory on each, but again, if you have more than that on each node then that's even better.

It's important that the IP addresses for the nodes in your cluster do not change. If an IP of a cluster node changes, it can cause the cluster nodes to be unable to find each other. In *Chapter 10, Connecting to Networks*, we learned how to set up a static IP address, which is a good solution for preventing IP addresses from changing. As an alternative, you can set up a static lease in your DHCP server if you wish. It doesn't matter which solution you use, so long as you prevent the IP addresses of cluster nodes from changing.

For some readers, a more accessible method of setting up comes in the form of **MicroK8s**, which allows you to even set up Kubernetes on a single machine. If your only goal is to set up a simple test installation, it's one of the best and easiest methods of getting started. MicroK8s is not recommended for running a production cluster in an organization, but it's definitely a great way to learn. I do recommend that you work through the standard Kubernetes procedure with multiple nodes if you can, but in the next section we'll walk through setting up MicroK8s for those of you that would like to utilize that method.

Utilizing MicroK8s

If you don't have more than one machine or enough memory on your laptop or desktop to run multiple nodes inside virtualization software such as VirtualBox, MicroK8s is a simple way to set up a Kubernetes instance for testing the platform, as well as going through the examples in this chapter. MicroK8s is actually provided by Canonical, the makers of Ubuntu. That just goes to show you how important Kubernetes is to the Ubuntu distribution, its own creator is going the extra mile to contribute to the platform. MicroK8s is available for Linux, macOS, as well as Windows. So regardless of which operating system you're running on your laptop or desktop, you should be able to install and use it. If nothing else, it gives you a great test installation of Kubernetes that will come in handy as you learn.

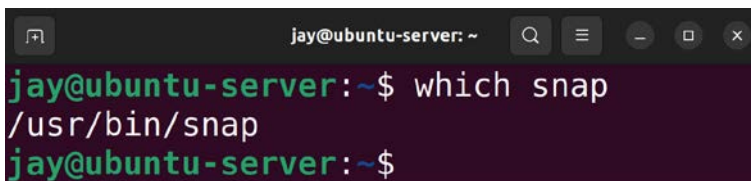
To set it up, follow along with one of the subsections below that matches the operating system installed on your computer.

Installing MicroK8s on Linux

When it comes to Linux, MicroK8s is distributed as a snap package. We covered snap packages back in *Chapter 3, Managing Software Packages*, and it's a great solution for cross-distribution package management. If you run a recent version of Ubuntu on your computer, then you should already have support for snap packages and you can proceed to install MicroK8s. If you're running a distribution of Linux on your computer other than Ubuntu, then you may not have access to the snap command by default. If in doubt, you can use the `which` to see if the command is available:

```
which snap
```

If you do have the ability to install snap packages, then your output should be similar to the following:

A terminal window screenshot showing a command prompt. The prompt is 'jay@ubuntu-server: ~'. The user enters the command 'which snap'. The output is '/usr/bin/snap'. The prompt returns to 'jay@ubuntu-server: ~\$'.

```
jay@ubuntu-server: ~$ which snap
/usr/bin/snap
jay@ubuntu-server: ~$
```

Figure 18.1: Checking if the snap command is available

If you see no output when you run `which snap`, then that means your distribution doesn't have support for this package type installed. Canonical makes the snap command available to distributions outside of Ubuntu, so if you're using a distribution other than Ubuntu, it's usually just a matter of installing the required package.

Canonical has additional information available on the following site, which shows the process of enabling snap for several distributions: <https://snapcraft.io/docs/installing-snapd>.

For example, the documentation on that site gives the following command to install the required package for Fedora:

```
sudo dnf install snapd
```

Essentially, as long as you follow along with the instructions for setting up snap that are specific to the distribution of Linux you're running, the process should be simple enough. If for some reason your distribution isn't supported, you can simply use the same Ubuntu installation as you've been using during this book, which will have snap support built in.

Once you either confirm you already have support for snap on your computer, or you successfully enable the feature, you can install MicroK8s with the following command:

```
sudo snap install microk8s --classic
```

If successful, you should see a confirmation in your terminal that the process was successful:

A terminal window titled 'jay@ubuntu-server: ~' showing the command 'sudo snap install microk8s --classic' being executed. The output shows the password prompt, the command execution, and a confirmation message: 'microk8s (1.24/stable) v1.24.0 from Canonical ✓ installed'. The prompt returns to 'jay@ubuntu-server:~\$'.

Figure 18.2: Setting up MicroK8s on a Linux installation

Now that we have MicroK8s installed on your Linux computer, we can proceed through the chapter. Or, you can check out the next section to see the installation process for macOS.

Installing MicroK8s on macOS

On macOS, the installation process for MicroK8s is similar. The process for Mac will utilize **Homebrew**, which is an addon for macOS. Homebrew isn't specific to Kubernetes or MicroK8s; it gives you the ability to install additional packages on your Mac that aren't normally available, with MicroK8s being one of many. Homebrew is essentially a command-line utility with syntax similar to Ubuntu's apt command.

To install Homebrew, visit <https://brew.sh> in your browser, and the command required to set it up will be right there on the site. I could insert the command to install Homebrew right here on this page, but the process may change someday, so it's better to get the command right from the official website for the utility. At the time of writing, the page looks like this:

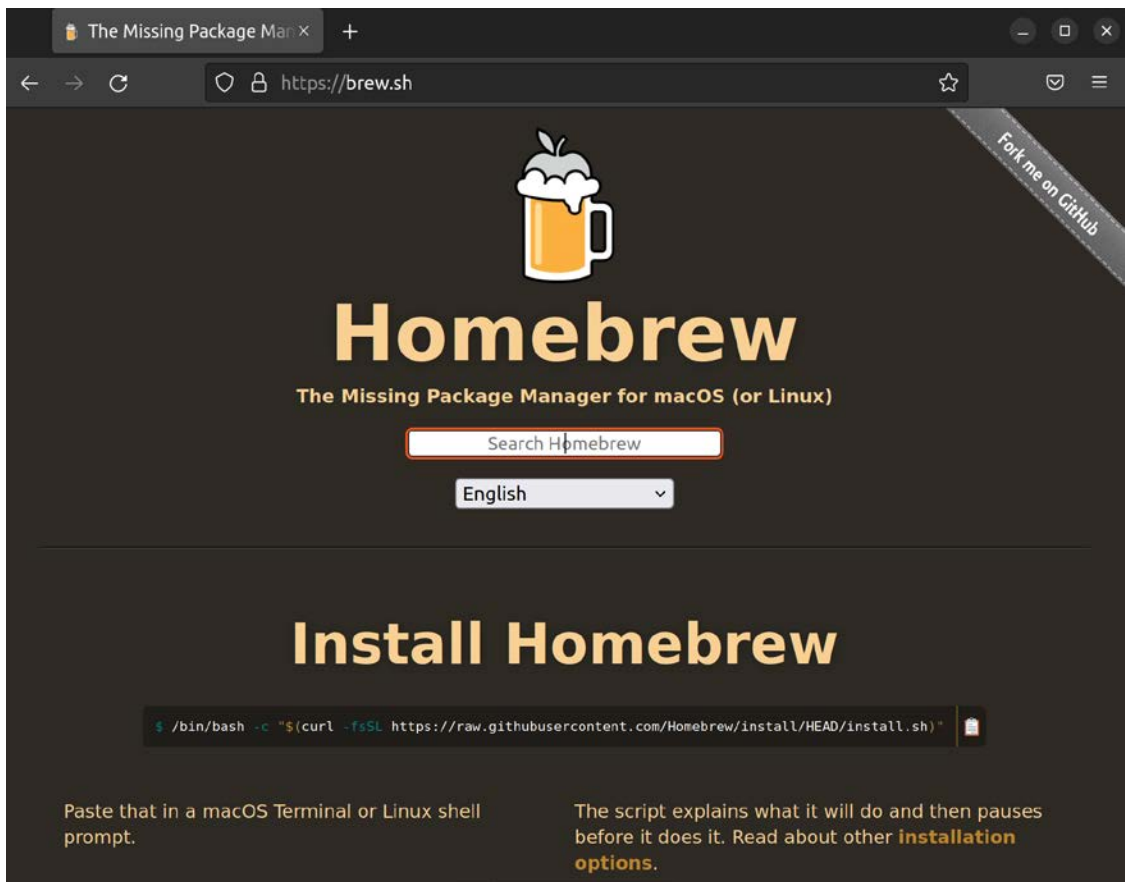


Figure 18.3: The Homebrew website

In the screenshot, you'll notice the command with white text on a black highlight. It's cut off a bit, due to the command being wider than this page. You should see the entire command on the site, so you can copy it from there and paste it into your Mac's terminal.

It's becoming increasingly popular for application developers to provide a command on their website to install their application that you paste directly into your terminal. This is very convenient and allows you to set up an app quickly. But you should always research and inspect such commands before pasting them into your terminal (regardless of your operating system). It's possible that an outside actor could hijack the command on a website to trick you into running something nefarious. You can use the `wget` command to download the script their command will end up running, so you can check it to ensure it's not doing something evil. Since this method of deploying software is getting more and more common, I recommend getting into the habit of checking commands before running them (especially if they use `sudo`).

Once you have Homebrew installed on your Mac, you can proceed to install MicroK8s with Homebrew with the following command:

```
brew install ubuntu/microk8s/microk8s
```

Once that command finishes, you should have MicroK8s installed on your Mac. In the next sub-section, we'll see the same process in Windows.

Installing MicroK8s on Windows

If you'd like to try out MicroK8s on a laptop or desktop computer running Windows 10, there's a specific installer available that will allow you to get it up and running. On the MicroK8s website, at <https://microk8s.io>, you should see a large green button labeled **Download MicroK8s for Windows**, and if you click on it, you'll be able to download the installer:

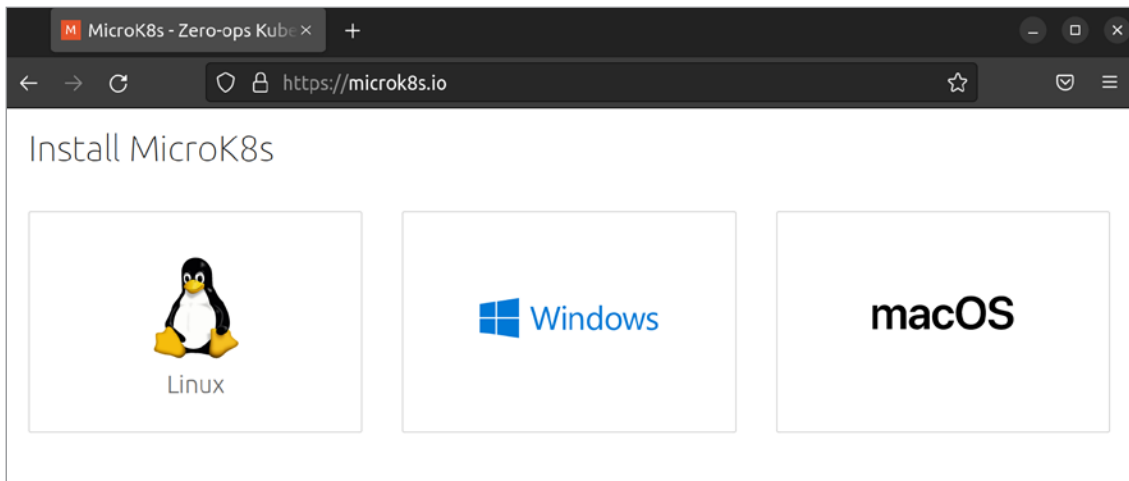


Figure 18.4: The MicroK8s website, with a button to download the installer for Windows

Once the download finishes, you can then launch the installer, which will have several different sections, all of which you can accept the defaults for. Click **Next** and then the **Install** button to begin the actual installation:

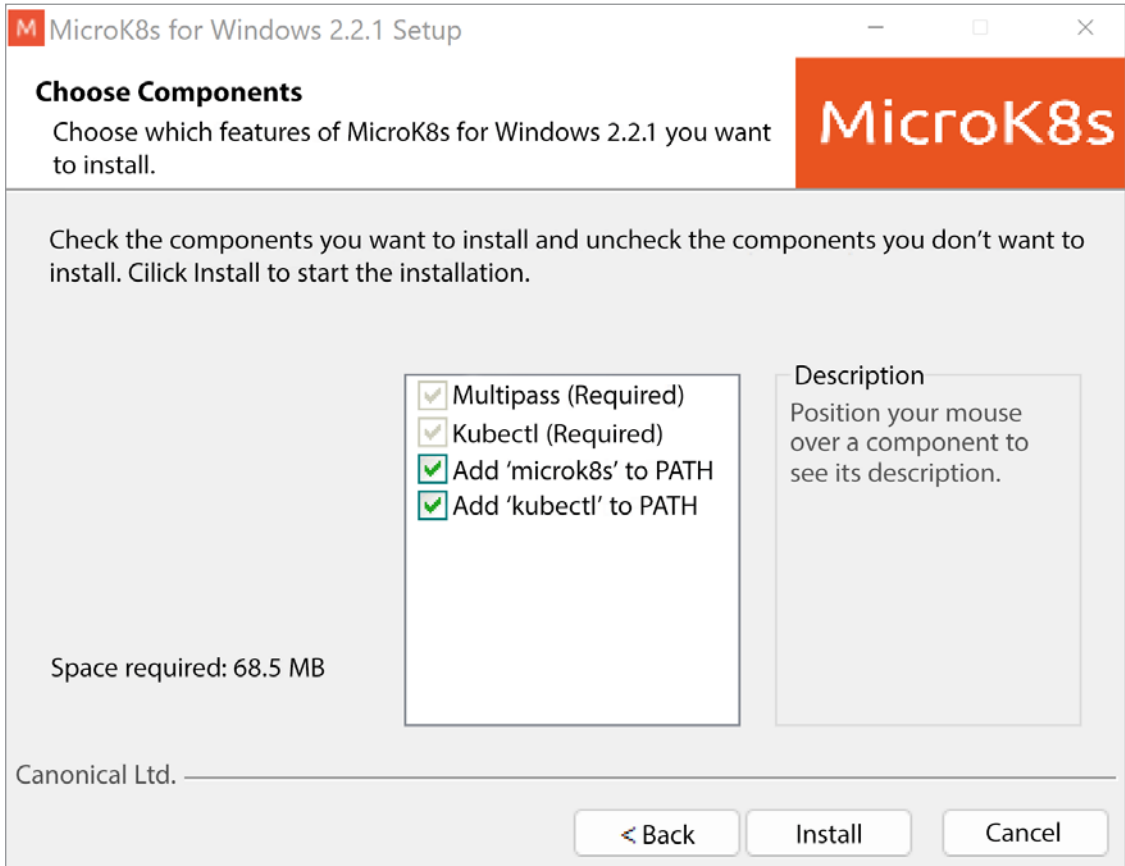


Figure 18.5: The MicroK8s installer for Windows

After the installation begins, another installation will also launch in parallel that will ask you to select a hypervisor:

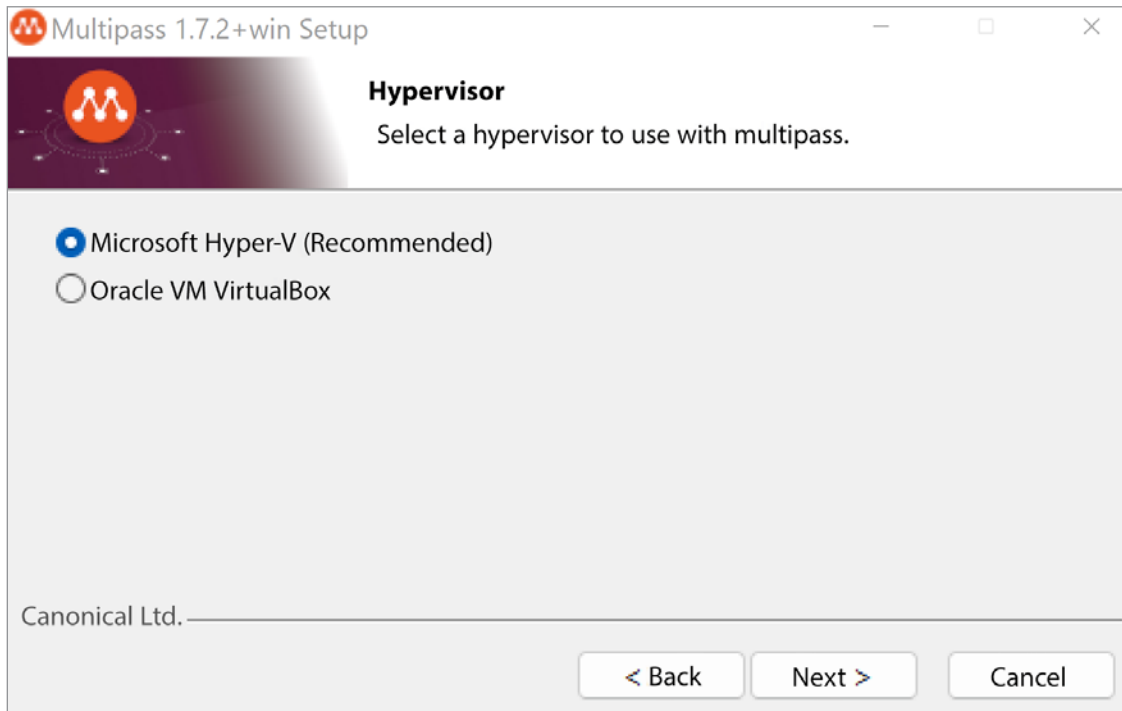


Figure 18.6: The MicroK8s installer for Windows, selecting a hypervisor

For this, you can leave the default selection on Microsoft Hyper-V, and click **Next**. You can keep the remaining options at their defaults and click **Next** through any remaining prompt that comes up. Depending on the setup of your computer, it may require you to reboot in order to finalize the required components before the installation can be completely finished. If you do see a prompt to reboot, do so, and then launch the installer again.

You'll see the following window appear, asking you to set up the parameters for MicroK8s, specifically how many virtual CPUs to utilize, how much RAM to provide, disk space, and so on:

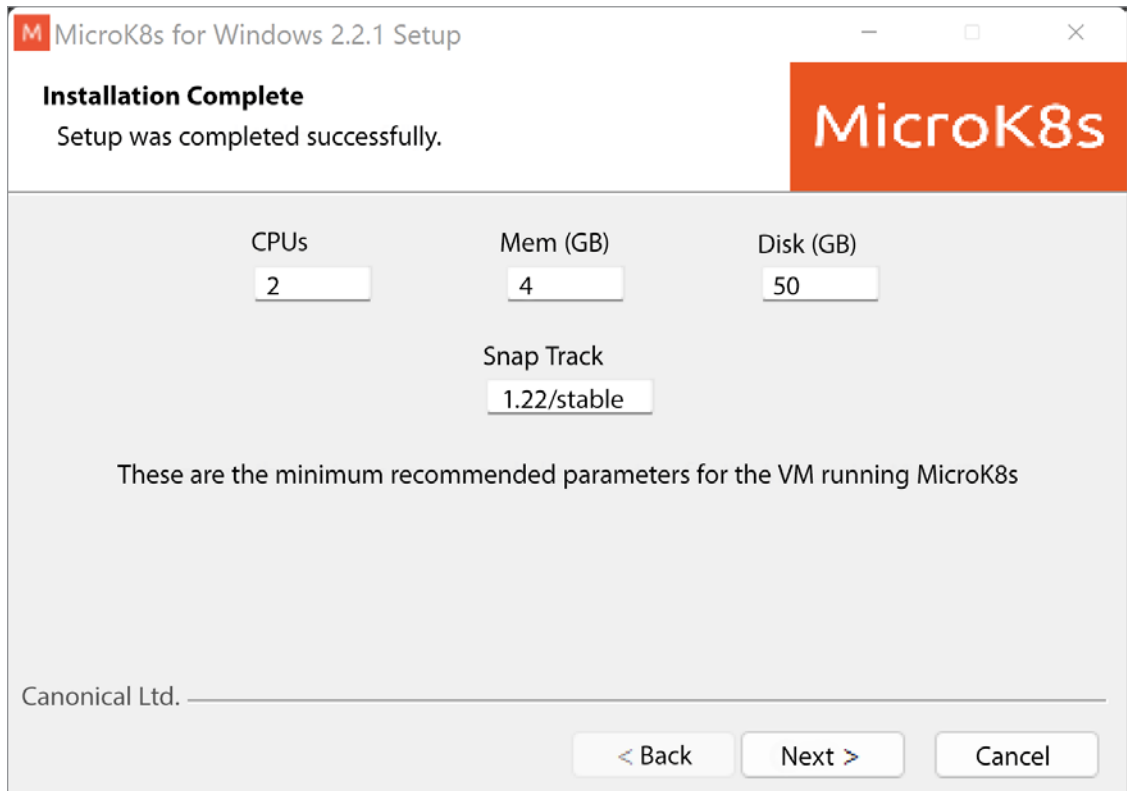


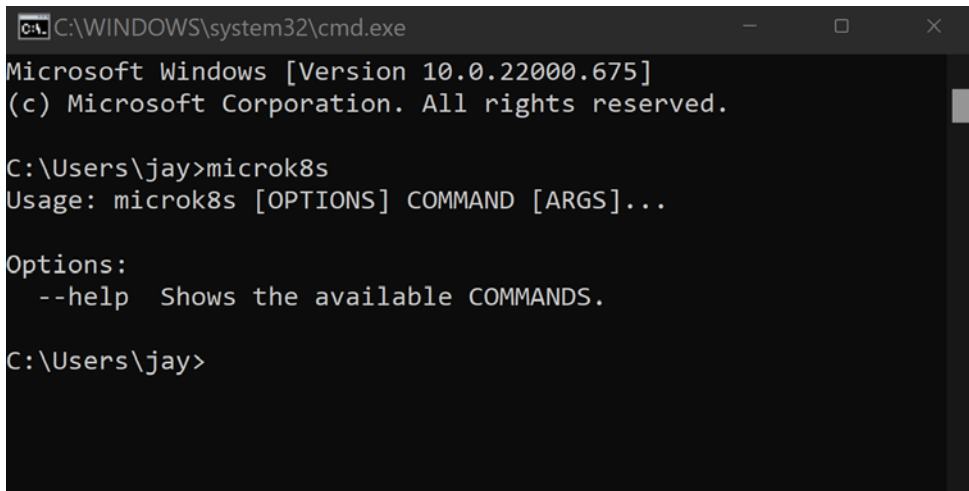
Figure 18.7: The MicroK8s installer for Windows, allocating resources

You can simply choose the defaults for this screen as well and click **Next**. Once the process is complete, you can click the **Finish** button to exit the installer.

Interacting with MicroK8s

At this point, if you've decided to utilize MicroK8s, you should have it installed on your computer. How you actually interact with it depends on your operating system. In each case, the `microk8s` command (which we'll cover shortly) is used to control MicroK8s. Which app you use in order to enter `microk8s` commands differs from one platform to another.

On Windows, you can use the Command Prompt app, which is built into the operating system. The `microk8s` command is recognized and available for use after you install MicroK8s. When you enter the `microk8s` command by itself, it should display basic usage information:

A screenshot of a Windows Command Prompt window. The title bar shows the path 'C:\WINDOWS\system32\cmd.exe'. The window content displays the following text:

```
Microsoft Windows [Version 10.0.22000.675]
(c) Microsoft Corporation. All rights reserved.

C:\Users\jay>microk8s
Usage: microk8s [OPTIONS] COMMAND [ARGS]...

Options:
  --help  Shows the available COMMANDS.

C:\Users\jay>
```

Figure 18.8: Executing the `microk8s` command with no options in a Windows Command Prompt

With macOS, you can use the built-in terminal app that's provided with the operating system. You may have another step to complete though before you can actually use MicroK8s. If you enter the `microk8s` command, and you receive an error informing you that support for `multipass` needs to be set-up, then you can run the following command to fix it:

```
microk8s install
```

If you're curious, **Multipass** is a technology that's also created by Canonical that allows you to quickly set up an Ubuntu instance for testing. It's not specific to MicroK8s or even Kubernetes, but in our case it's used in the background to facilitate MicroK8s. Multipass is not covered in this book, but it's worth taking a look at if you think you'll benefit from the ability to quickly set up Ubuntu instances to test applications and configurations on. It's available for all of the leading operating systems.

When it comes to Linux, if you've already gone through the process of setting up MicroK8s, you should be ready to use it immediately. Open up your distribution's terminal app and try the `microk8s` command to see if it's recognized. If it is, you're ready to move on.

To check the status of MicroK8s, the following command can be used to give you an overview of the various components. Before you run it, note the inclusion of `sudo` at the beginning. By default, this is required if your underlying operating system is Linux.

If you're running Windows 11 or macOS, you shouldn't need `sudo`. This difference has to do with how the underlying operating system handles permissions, which is different from operating system to operating system. So, if you're not using Linux on your PC, feel free to omit `sudo` from `microk8s` commands:

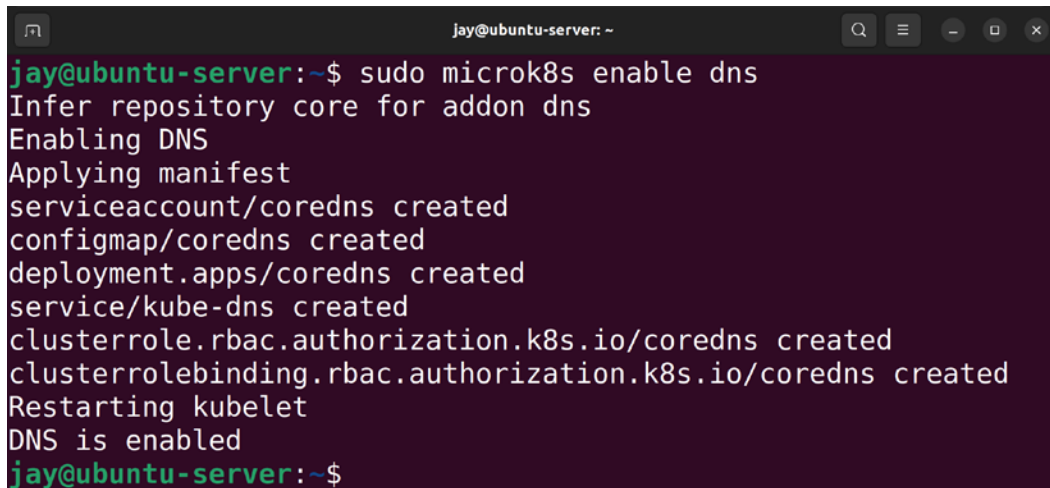
```
sudo microk8s kubectl get all --all-namespaces
```

Don't worry about the details regarding the individual components of Kubernetes at the moment, we'll cover what you need to know as we go through the rest of the chapter. For now, so long as you don't see errors when you run the previous command and check the status, you should be in good shape to continue.

By default, our MicroK8s installation comes with only the addons that are required for it to run. There are other addons we can enable, such as `storage`, which gives us the ability to expose a path on the host (the underlying operating system) to Kubernetes; `gpu`, which allows us to utilize NVIDIA GPUs within the containers; and others. We don't need to worry about these for now, but we should at least enable the `dns` addon, which sets up DNS within our cluster. It's not necessarily required, but not having it can create issues down the road when it comes to name resolution, so we may as well enable it now:

```
sudo microk8s enable dns
```

You should see output similar to the following:

A terminal window titled 'Jay@ubuntu-server: ~' with search, menu, and window control icons in the title bar. The terminal shows the command 'sudo microk8s enable dns' being executed. The output is as follows:

```
Jay@ubuntu-server:~$ sudo microk8s enable dns
Infer repository core for addon dns
Enabling DNS
Applying manifest
serviceaccount/coredns created
configmap/coredns created
deployment.apps/coredns created
service/kube-dns created
clusterrole.rbac.authorization.k8s.io/coredns created
clusterrolebinding.rbac.authorization.k8s.io/coredns created
Restarting kubelet
DNS is enabled
Jay@ubuntu-server:~$
```

Figure 18.9: Enabling the DNS addon in MicroK8s on a Linux installation

In my tests, I find that the command to enable the addon seems to stop responding for a while, with no output. Don't close the window; it should catch up and then display information about the process of installing the addon eventually. If this happens, just wait it out a bit.

With the previous command, we've enabled the `dns` addon. As I've mentioned, there are other addons available, but that's enough for now. At this point, just keep in mind that you can extend MicroK8s with addons, so it may be worth exploring in more detail later on if you wish. A list of addons is included in a link at the end of this chapter.

As mentioned earlier, if you're using Linux to run MicroK8s, then you would've used `sudo` with the `microk8s` command we used earlier. If you'd like to remove the requirement of using `sudo` on Linux, you can do so by adding your user to the `microk8s` group. That can be done with the following command:

```
sudo usermod -a -G microk8s <yourusername>
```

After you log out and then log in again, you should be able to run `microk8s` commands without `sudo`.

As we'll discover as we proceed through the rest of the chapter, the `kubectl` command is generally used to interact with a Kubernetes cluster and manage it. `kubectl`, short for **Kube Control**, is the standard utility you use to perform many tasks against your cluster. We'll explore this more later on. But specific to MicroK8s, it's important to understand that it uses its own version of `kubectl` that's specific to it. So with MicroK8s, you would run `microk8s kubectl` instead of simply `kubectl` to interact with the cluster. Having a separate implementation of `kubectl` with MicroK8s allows you to target an actual cluster (the `kubectl` command by itself) or specifically your installation of MicroK8s (prefix `kubectl` commands with `microk8s`), so one won't conflict with the other. As we work through the chapter, I'll call out `kubectl` by itself when we get to the actual examples, so it will be up to you to remember to use `microk8s` in front of such commands as needed.

Now that we have our very own installation of MicroK8s on our laptop or desktop, we can proceed through the examples in this book. You can skip ahead to the *Deploying containers via Kubernetes* section later in this chapter to begin working with Kubernetes. In the next section though, we'll take a look at setting up a Kubernetes cluster manually without MicroK8s, which is closer to the actual process you'd use in an actual production implementation.

Setting up a Kubernetes cluster

In the previous section, we set up MicroK8s, which provides us with a Kubernetes cluster on a single machine, which is great for testing purposes. That might even be all you need in order to learn Kubernetes and see how it works. If you can, I still recommend setting up a cluster manually, which will give you even more insight into how the individual components work together. That's exactly what we're going to do in this section.

Before we do get started, it's important to synchronize our mindset a bit. Of all of the activities we've worked through so far, setting up a Kubernetes cluster manually is easily the most complex. Kubernetes itself is made up of many components, as well as settings. If any one component is incorrect or a setting is misconfigured, the entire process can fail. In this section, a great deal of care and attention was spent to ensure (as much as possible) that the process works to the point where it's completely reproducible and has been broken down to only the required components to simplify everything. However, if the process doesn't work out well when you go to try it, don't worry - it's perfectly fine if we encounter an issue. We're still learning, and fixing things is the best way to learn. So the bottom line is to have fun, and be patient.

What does a typical Kubernetes cluster look like? When it comes to a production installation in an actual data center, having Kubernetes installed on multiple servers is commonplace. Typically, one of them will act as the controller node, and then you can add as many worker nodes as you need.

As your needs expand, you can add additional servers to provide more worker nodes to your cluster. Setting up a controller Kubernetes node and then individual workers is a great way to see the actual relationship in action. And that's exactly what we're going to do in this section.

As I walk you through the process, I'm going to do so utilizing three VMs. The first will be the controller, and the remaining two will be workers. On your end, it doesn't really matter how many workers you decide to go with. You can have a single worker, or a dozen—however many you'd like to set up is fine. If you do set up more nodes than I do, then you would simply repeat the commands in this section for the additional nodes. For the nodes on your end, feel free to use whatever combination of physical or VMs makes sense and fits within the resources of the equipment you have available.

Before we get started, you'll want to have already installed Ubuntu Server on each machine and install all of the updates. It's also a good idea for you to create a user for yourself if you haven't already done so. The remainder of this section will assume you've already set up Ubuntu on each. After you've installed Ubuntu on everything, it's a good idea to also configure the hostnames on each node as well to make it easier to tell them apart. For example, on the node that I'm intending to use as the controller, I'm going to set the hostname as `controller`. For the workers, I'll name them `node-01` and `node-02` respectively. You can name yours in whatever theme makes sense, but the reason I specifically mention it is to make sure that you name them something, as by default they'll each show the same hostname in the command prompt, and that may be confusing.

Now that you have everything you need, let's set up Kubernetes!

Preliminary setup

In this section, we're going to complete some miscellaneous setup tasks before we actually start building our cluster. These are basically the prerequisites for our cluster to function.

We should first ensure that each of the servers we intend on using for the cluster has either a static lease, or a static IP that will not change. You can change your server's IP address at any time, but with Kubernetes, there's some additional steps involved if you do change the IP later. For this reason, I suggest setting your static lease or static IP right now, *before* we build the cluster. That way, you won't have to worry about that later. The process was covered in *Chapter 11, Setting Up Network Services*, so I won't repeat that here. On your side, be sure to set up a static lease or static IP, and then continue further with our setup here.

Next, if the devices you've selected for this exercise includes servers that are being built using the Raspberry Pi, then there are some special requirements that are specific to the Pi. If you are using one or more Raspberry Pi units, be sure to complete this step (otherwise, skip this if you're NOT using the Raspberry Pi).

To set boot parameters specific to the Raspberry Pi, open up the following file in your editor of choice:

```
sudo nano /boot/firmware/cmdline.txt
```

What you're about to do in this file, is add some cgroup options to control how Kubernetes is able to interact with the hardware of your device. You'll want to add the following options to the existing line within the file. Don't create a new line. Instead, add these options to the very end of the first (and only) line within that file:

```
cgroup_enable=cpuset cgroup_enable=memory cgroup_memory=1 swapaccount=1
```

Save and exit the file, and that should be the only change we'll be making that's specific to the Raspberry Pi.

In order to build a Kubernetes cluster, we'll need to set up something called a **Container Runtime**. A Container Runtime is the means by which Kubernetes will run containers within the cluster. Without a runtime, there's no ability for Kubernetes to run even a single container. While Kubernetes is a container orchestration solution, it doesn't mandate which container runtime you use. The suggested runtime known as **containerd**, so that's what we'll use in our cluster. Other container runtimes include (but aren't limited to) Docker, CRI-O, and there's others. We've already worked with Docker in this book, but for our cluster, we're going to deploy it with the suggested containerd runtime.

When it comes to the containers we can run, there's no difference - containerd for example can run Docker containers just like we've been doing. So even though we're going to go with a different container runtime, it shouldn't really make a difference at all when it comes to the end result.

In this section, the commands that I'm going to have you run should be executed on each of the instances you plan on using with Kubernetes, regardless of whether or not you're working with the intended controller or a node; these commands need to be run on each. There will be commands specific to the controller and the nodes later on, but I'll mention the intended target if it's something I'd like you to do on specifically one or the other.

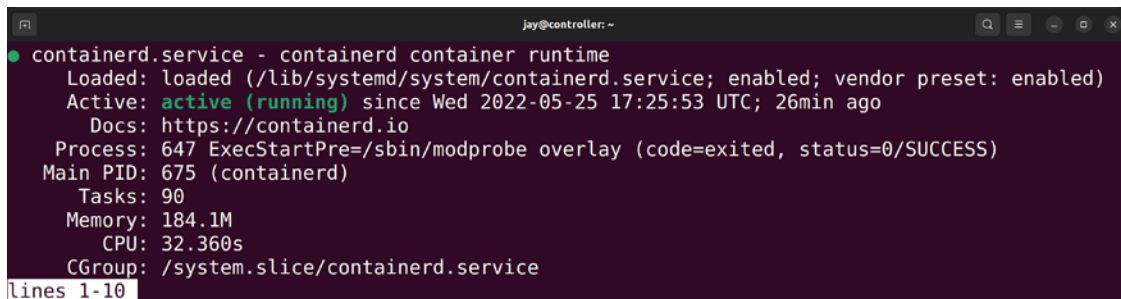
The first thing we're going to do is install the containerd package:

```
sudo apt install containerd
```

After the package is installed, we'll check the status of it and ensure that it's running:

```
systemctl status containerd
```

We should see that the unit is running:



```
jay@controller: ~  
● containerd.service - containerd container runtime  
   Loaded: loaded (/lib/systemd/system/containerd.service; enabled; vendor preset: enabled)  
   Active: active (running) since Wed 2022-05-25 17:25:53 UTC; 26min ago  
     Docs: https://containerd.io  
   Process: 647 ExecStartPre=/sbin/modprobe overlay (code=exited, status=0/SUCCESS)  
   Main PID: 675 (containerd)  
     Tasks: 90  
    Memory: 184.1M  
       CPU: 32.360s  
   CGroup: /system.slice/containerd.service  
lines 1-10
```

Figure 18.10: Checking the status of the containerd runtime

We're not quite done with `containerd`, though. It's going to need some default configuration in order to function properly, but by default, there's no sample config provided automatically. Let's create that config, starting with adding a directory to our server to house the configuration file:

```
sudo mkdir /etc/containerd
```

Next, we'll create the config file for `containerd` with the following command:

```
containerd config default | sudo tee /etc/containerd/config.toml
```

That command will not only create the default configuration file for `containerd`, it also prints the configuration information to the screen as well so we can see what the default values are. However, while most of the default settings are fine, we'll need to make some adjustments to this file. Open up the `/etc/containerd/config.toml` file in your editor of choice.

The first change to make to this file, is to set the `cgroup` driver to `systemd`. To do so, look for the following line:

```
[plugins."io.containerd.grpc.v1.cri".containerd.runtimes.runc.options]
```

Several lines below that, you'll see the following line:

```
SystemdCgroup = false
```

Change that line to `true` instead:


```
SystemdCgroup = true
```

At this point, we're done with `containerd` for now. Let's move on and configure some important system tweaks. The first of these is disabling swap. Although it's usually not a good idea to run a server without swap, setting up a Kubernetes cluster is the exception to this. In fact, when we go to initialize the cluster later, the process will actually abort if swap is enabled. So let's take care of disabling it now.

First, we'll run the following command to disable swap:

```
sudo swapoff -a
```

After doing this, if we run `free -m` we should see all zero's for swap:

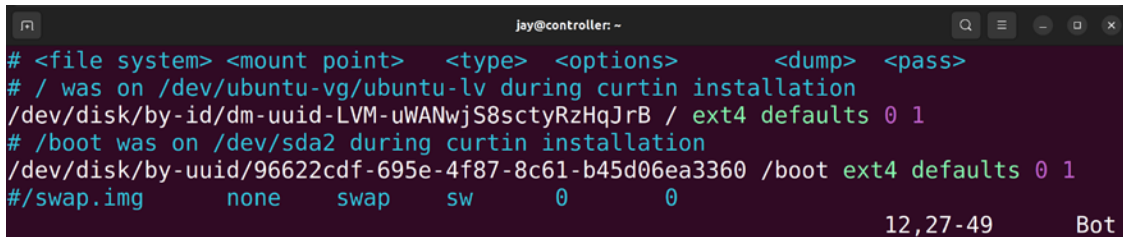


```
jay@controller: ~  
jay@controller:~$ free -m  
              total        used         free       shared  buff/cache   available  
Mem:           1977         635          440            2          901       1185  
Swap:            0            0            0
```

Figure 18.11: Running the `free -m` command to check that swap is disabled

However, the command that we've just run to disable swap was not a permanent change. When we reboot our server, swap will be automatically re-enabled. In order to disable swap for good, we'll need to edit the `/etc/fstab` file and comment out the line that activates swap during boot.

The following screenshot shows an example configuration line for swap, but it's been commented out by inserting the # character at the beginning of that line:



```

jay@controller: ~
# <file system> <mount point> <type> <options> <dump> <pass>
# / was on /dev/ubuntu-vg/ubuntu-lv during curtin installation
/dev/disk/by-id/dm-uuid-LVM-uWANwjS8sctyRzHqJrB / ext4 defaults 0 1
# /boot was on /dev/sda2 during curtin installation
/dev/disk/by-uuid/96622cdf-695e-4f87-8c61-b45d06ea3360 /boot ext4 defaults 0 1
#/swap.img none swap sw 0 0
12,27-49 Bot

```

Figure 18.12: The `/etc/fstab` file, with the swap line commented out

At this point, swap should be turned off, and considering we've commented out the swap line in the `/etc/fstab` file, swap won't be re-enabled the next time we start our server.

Next, we'll need to make a change to the `/etc/sysctl.conf` file to enable bridging. This is similar to what we worked through as we set up an internet gateway back in *Chapter 11, Setting Up Network Services*. Once we've opened up the `/etc/sysctl.conf` file in an editor, we'll need to uncomment the following line:

```
#net.ipv4.ip_forward=1
```

It should now look like this:

```
net.ipv4.ip_forward=1
```

With that line uncommented, bridging should be enabled the next time we start our server.

Next, we'll create the following file, which we'll use to ensure a required kernel module is loaded when the server starts:

```
sudo nano /etc/modules-load.d/k8s.conf
```

Inside that file, we'll add the following:

```
br_netfilter
```

That's actually it for that file, so save and exit the editor. The `br_netfilter` module assists with networking for our eventual Kubernetes cluster, and will need to be enabled in order for the cluster to function. By creating the `/etc/modules-load.d/k8s.conf` file, we're ensuring this kernel module is loaded automatically when we start our server.

Before we continue, further, let's take a moment and ensure we've done everything we need to do up to now. At this point, on each of the servers you intend to use with Kubernetes (the controller as well as the nodes) you should've accomplished the following:

- Installed all updates
- Set a hostname
- Set up a static IP or lease on each node
- Set Raspberry Pi-specific boot options (if you're using a Raspberry Pi)

- Installed `containerd`
- Created the `/etc/containerd/config.toml` file for `containerd` and also set the `cgroup` driver
- Disabled swap
- Added the `/etc/modules-load.d/k8s.conf` file with the `br_netfilter` module listed inside
- Edited the `/etc/sysctl.conf` file to enable bridging

All of the above should've been completed on every server you plan to use with your cluster. Once you're sure that you've performed each of those tasks, we should reboot each of our nodes:

```
sudo reboot
```

Once your nodes finish starting back up, we can proceed to actually install Kubernetes.

Installing Kubernetes

Now it's finally time to start building our cluster, and we'll start by installing the required packages. To do so, we'll need to add the appropriate repository so we can have access to the required packages for installing Kubernetes. To add the repository, we'll first add the key for the repository so our server will accept it as a trusted resource:

```
sudo curl -fsSLo /usr/share/keyrings/kubernetes-archive-keyring.gpg https://packages.cloud.google.com/apt/doc/apt-key.gpg
```

To add the actual repository itself, we will run the following command:

```
echo "deb [signed-by=/usr/share/keyrings/kubernetes-archive-keyring.gpg] https://apt.kubernetes.io/ kubernetes-xenial main" | sudo tee /etc/apt/sources.list.d/kubernetes.list
```

After adding the repository, we'll update our local index:

```
sudo apt update
```

What you may notice is that the repository URL references `xenial` instead of the actual codename for Ubuntu 22.04, which is `jammy` (short for Jammy Jellyfish). At the time of writing, there is no dedicated Kubernetes repository for Ubuntu 22.04 yet, but the process will still work just fine. It's possible that by the time you're reading this, there will be a dedicated repository for 22.04. But for now, we can use the line mentioned above for our repository file.

Next, we can install the packages required for Kubernetes:

```
sudo apt install kubeadm kubectl kubelet
```

We're installing three packages, `kubeadm`, `kubectl`, and `kubelet`:

- **kubeadm:** The `kubeadm` package gives us tools we can use to “bootstrap” our cluster. We can use this tool to initialize a new cluster, join a node to an existing cluster and upgrade the cluster to a newer version.
- **kubectl:** This package provides us with the Kubernetes command-line tool, `kubectl`. We will use this tool to interact with our cluster and manage it.

- **kubelet:** The Kubernetes kubelet acts as an agent that facilitates communication between nodes. It also exposes API endpoints that can be used to enable additional communication and features.

Unlike all of the previous commands we've run through so far while setting up our cluster, the following will be entered and ran only on the node that you've designated as the controller (before you run it, read the paragraph that follows to know how to customize the command):

```
sudo kubeadm init --control-plane-endpoint=172.16.250.216 --node-name
controller --pod-network-cidr=10.244.0.0/16
```

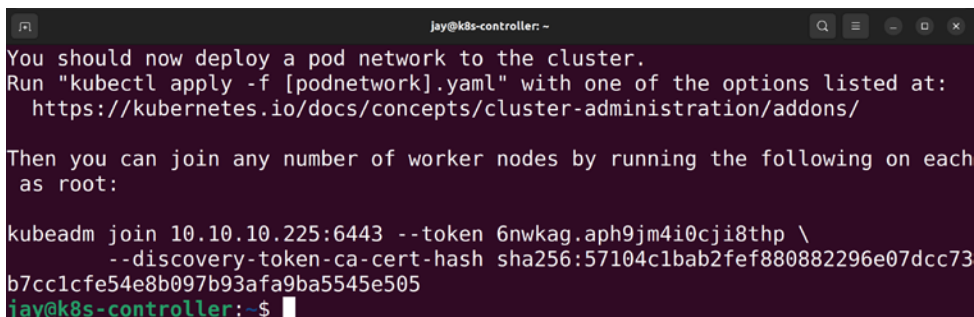
The previous `kubeadm init` command should be customized a bit, before you run it. There's a few things you should change to ensure they match your environment. In fact, I bolded the individual settings you should customize. I'll also describe some additional details about those particular settings now.

`--control-plane-endpoint=172.16.250.216`: For this option, the IP address that is mentioned here should be the same IP address that's assigned to your server. I filled in the value I used for my test server, but you should make sure this matches the actual IP on your side. Since this is a very specific setting, this is one of the reasons why I recommended you finalize your IP address for your nodes *before* you set them up in the cluster.

`--node-name controller`: When I set up my test server, I simplified its hostname down to simply `controller`. On your side, you can set this to match your controller's hostname as well.

Note that I didn't bold the second IP address in the command, `10.244.0.0/16`. The reason for this, is that you should not change that particular IP declaration. That's actually the IP scheme for the **Pod Network**, which is an internal network to Kubernetes and is not accessible from the outside. You can actually customize this to your own scheme, but then you'd have to change other settings as well. Since this is a dedicated internal network, there shouldn't be any need to change this so I recommend to leave it as-is.

The `kubeadm init` command that I mentioned earlier, after you customize it, will initialize the cluster and assign it a Pod Network. If the initialization process is successful, you'll see a `join` command printed to the terminal at the end. If you do see it, then congratulations! You've successfully set up a Kubernetes cluster. It's not a very usable cluster yet, because we haven't added any worker nodes to it at this point. But you do, by definition, have a Kubernetes cluster in existence at this point. On my controller node, I see the following output:



```

jay@k8s-controller: ~
You should now deploy a pod network to the cluster.
Run "kubectl apply -f [podnetwork].yaml" with one of the options listed at:
  https://kubernetes.io/docs/concepts/cluster-administration/addons/

Then you can join any number of worker nodes by running the following on each
as root:

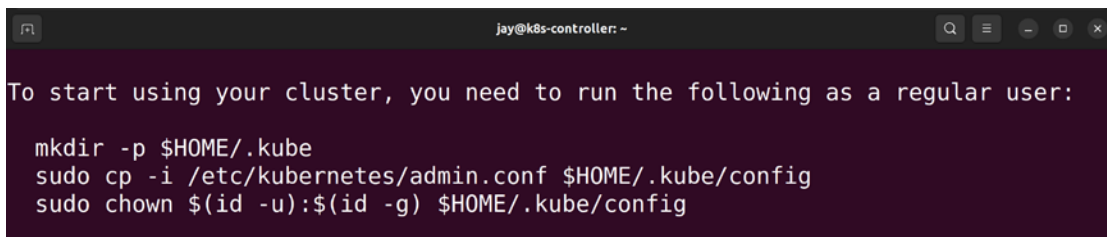
kubeadm join 10.10.10.225:6443 --token 6nwkag.aph9jm4i0cji8thp \
--discovery-token-ca-cert-hash sha256:57104c1bab2fef880882296e07dcc73
b7cc1cfe54e8b097b93afa9ba5545e505
jay@k8s-controller: $

```

Figure 18.13: Successful initialization of a Kubernetes cluster, showing a join command

The join command that's shown to you after the process is complete is very special—you can copy that command and paste it into the command prompt of a worker node to join it to the cluster, but don't do that just yet. For now, copy that command and store it somewhere safe. You don't want to allow others to see it, because it contains a hash value that's specific to your cluster and allows someone to join nodes to it. Technically, showing my join command with the hash value is the last thing I should do in a book that will be seen and read by many people, but since this is just a test cluster with no actual value, I don't mind you seeing it.

Also, in the same output as the join command, are three additional commands we should run on the controller node. If you scroll up in your terminal window to before the join command, you should see output similar to the following:



```

jay@k8s-controller: ~
To start using your cluster, you need to run the following as a regular user:

mkdir -p $HOME/.kube
sudo cp -i /etc/kubernetes/admin.conf $HOME/.kube/config
sudo chown $(id -u):$(id -g) $HOME/.kube/config

```

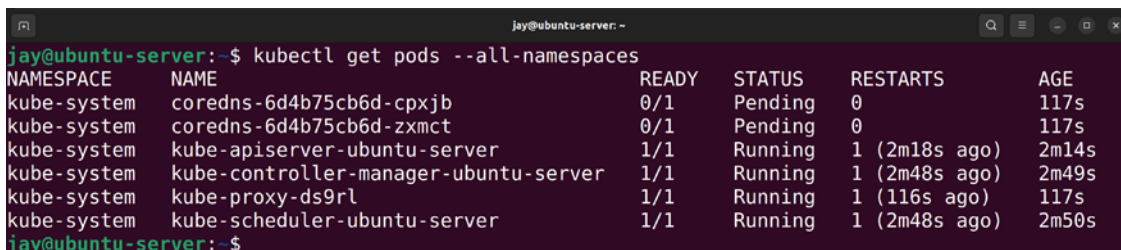
Figure 18.14: Output of the initialization process for Kubernetes showing additional commands to run

For those three commands, you can copy and paste them as-is right back into the terminal, one by one. Go ahead and do that. Essentially what you're doing is creating a local config directory for `kubectl` right in your user's home directory. Then, you're copying the `admin.conf` file from `/etc/kubernetes` and storing it in the newly created `.kube` folder in your home directory with a new name of `config`. Finally, you're changing ownership of the `.kube` directory to be owned by your user. This will allow your user account to manage Kubernetes with the `kubectl` command, without needing to be logged in as root or use `sudo`.

Kubernetes itself consists of multiple **Pods**, within which your containers will run. We haven't deployed any containers yet, we'll do that later. But even though we didn't deploy any containers ourselves, there are some that are actually running already. See for yourself:

```
kubectl get pods --all-namespaces
```

On my end, I see the following:



```

jay@ubuntu-server: ~
jay@ubuntu-server:~$ kubectl get pods --all-namespaces
NAMESPACE   NAME                                     READY   STATUS    RESTARTS   AGE
kube-system  coredns-6d4b75cb6d-cpxjb              0/1     Pending  0           117s
kube-system  coredns-6d4b75cb6d-zxmct              0/1     Pending  0           117s
kube-system  kube-apiserver-ubuntu-server          1/1     Running  1 (2m18s ago)  2m14s
kube-system  kube-controller-manager-ubuntu-server 1/1     Running  1 (2m48s ago)  2m49s
kube-system  kube-proxy-ds9rl                      1/1     Running  1 (116s ago)   117s
kube-system  kube-scheduler-ubuntu-server          1/1     Running  1 (2m48s ago)  2m50s
jay@ubuntu-server:~$

```

Figure 18.15: Checking the status of pods running in our Kubernetes cluster

Notice that the namespace for each of these Pods is `kube-system`. This is a special namespace, where Pods related to the Kubernetes cluster itself will run. We didn't explicitly ask for these to run, they're run as part of the cluster and provide essential functionality. Another important column is `STATUS`. In the screenshot, we see that most of them are `Running`. The first two, though, are in a state of `Pending`, which means that they're not quite ready yet. This is actually expected, we haven't deployed an overlay network yet, which is required for the `coredns` pods to function.



Overlay networks are virtual networks that are created to serve as a network on top of another network. The concept is not specific to Kubernetes, but when used in Kubernetes, the overlay network manages communication between nodes.

Other Pods may also show a status of `Pending`, and those should automatically switch to `Running` when they finish setting themselves up. How long this takes depends on the speed of your hardware, but it should only take a few minutes.

The others will eventually change to a state of `Running` once their setup is finished – but they might also crash as well, and respawn. Since there's no overlay network yet, the pods aren't able to fully communicate yet. So if you see some errors, don't worry about that yet.

Let's deploy an overlay network so we can make the cluster more stable:

```
kubectl apply -f https://raw.githubusercontent.com/flannel-io/flannel/master/Documentation/kube-flannel.yml
```

When we check the status again, we should see an additional Pod running (`kube-flannel-ds`), and the Pods that had a state of `Pending` previously should now be `Running`:

```

jay@controller:~$ kubectl get pods --all-namespaces
NAMESPACE   NAME                                     READY   STATUS    RESTARTS   AGE
kube-system  coredns-6d4b75cb6d-4nw62              1/1     Running  0          2m2s
kube-system  coredns-6d4b75cb6d-cmtkv              1/1     Running  0          2m2s
kube-system  etcd-controller                        1/1     Running  0          2m16s
kube-system  kube-apiserver-controller              1/1     Running  0          2m16s
kube-system  kube-controller-manager-controller     1/1     Running  0          2m16s
kube-system  kube-flannel-ds-tnrkw                  1/1     Running  0          43s
kube-system  kube-proxy-qwx8h                       1/1     Running  0          2m2s
kube-system  kube-scheduler-controller              1/1     Running  0          2m16s
jay@controller:~$

```

Figure 18.16: Checking the status of Pods running in our Kubernetes cluster again

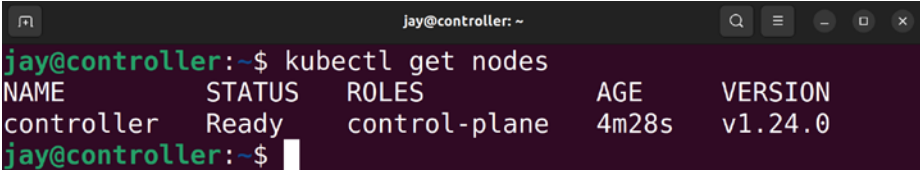
So, what exactly did we do? **Flannel** is a networking layer that can run on Kubernetes. Networking is required for Kubernetes to function, as Pods within the cluster need to be able to communicate with one another. There are multiple different types of networking models you can implement in Kubernetes, Flannel is just one available option. If you're using a cloud provider, such as AWS, then the networking model is typically chosen for you. Since we're building a cluster from scratch, we have to choose a networking model—Kubernetes itself doesn't come with one.

Setting up networking in a cluster that was manually created can be quite an involved task, Flannel is easy to set up (we simply deploy it) and its defaults meet the needs of Kubernetes and will get us up and running quickly. There are definitely some other options for the networking layer to consider for your cluster, but Flannel is good enough for us for what we need right now.

Next, it's time to watch the magic happen and join worker nodes to our cluster. We can check the status of all of our nodes with the following command:

```
kubectl get nodes
```

Right now though, we only have the controller showing up in the output since we haven't yet added any nodes:



```
jay@controller:~$ kubectl get nodes
NAME          STATUS    ROLES    AGE   VERSION
controller   Ready    control-plane  4m28s  v1.24.0
jay@controller:~$
```

Figure 18.17: Checking the status of the nodes within the cluster

To add a worker to our cluster, we can enter the `join` command we saw earlier on a node designated as a worker. If you recall, the `join` command was shown in the terminal when we first initialized our cluster. The command will look something like the following, which I've shortened a bit to fit on this page (I've added `sudo` to the beginning):

```
sudo kubeadm join 172.16.250.216:6443 --token zu5u3x.p45x0qkj137ine6i \
--discovery-token-ca-cert-hash sha256...1360c
```

You should consider the `join` command as private, and not show it to anyone, nor should you upload it to a Git repository or a documentation server. Reason being, it could be used by an outside actor to join something to your cluster. In my case, I truncated the hash so it's impossible to reproduce, but keep this in mind going forward.

For you, the command will be very different. The IP address in the command is for the controller, which will no doubt be different on your end. The hash value will be different as well. Basically, just copy the `join` command you were provided while initializing the cluster on the controller and paste it into each of your worker nodes. You should see a message in the output that the node was successfully added to the cluster:

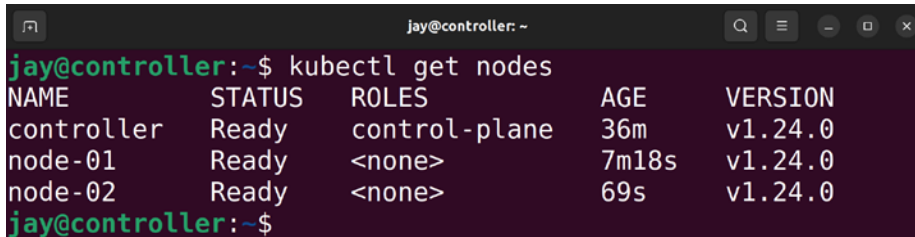


```
jay@node-01:~$ sudo kubeadm join 172.16.250.216:6443 --token zu5u3x.p45x0qkj137ine6i \
--discovery-token-ca-cert-hash sha256...1360c
This node has joined the cluster:
* Certificate signing request was sent to apiserver and a response was received.
* The Kubelet was informed of the new secure connection details.

Run 'kubectl get nodes' on the control-plane to see this node join the cluster.
jay@node-01:~$
```

Figure 18.18: Successfully adding a worker node to the Kubernetes cluster

After you've run the join command on each of your worker nodes (on however many you decided to create), you can run the `kubectl get nodes` command again on the controller and verify the new nodes appear on the list. I added two nodes, so I see the following output on my end:



```

jay@controller:~$ kubectl get nodes
NAME          STATUS    ROLES    AGE   VERSION
controller    Ready    control-plane   36m   v1.24.0
node-01       Ready    <none>    7m18s v1.24.0
node-02       Ready    <none>    69s   v1.24.0
jay@controller:~$

```

Figure 18.19: Output of `kubectl` showing worker nodes now added to the cluster

Once all of the nodes you plan on deploying show a STATUS of Ready, then your cluster setup is complete!

If you run into any trouble joining a node to the cluster, you can try to regenerate the join token. After a while, it's possible the original could've expired and the certificates won't be valid. To regenerate the join command, run the following from the controller node:

```
kubeadm token create --print-join-command
```

This will print a brand new join command you can use in place of the original.

At this point, our cluster exists and has one or more worker nodes ready to do our bidding. The next step is to actually use the cluster and deploy a container. That's exactly what we'll do in the next section.

Deploying containers via Kubernetes

Now it's time to see our work pay off, and we can successfully use the cluster we've created. At this point, you should have either set up MicroK8s, or manually created a cluster as we've done in the previous section. In either case, the result is the same: we have a cluster available that we can use to deploy containers.

Keep in mind that if you're using MicroK8s, you might need to prepend `microk8s` in front of `kubectl` commands, depending on how you set up MicroK8s. I'll leave it up to you to add `microk8s` to the front of such commands as you go along, if you're using MicroK8s and you don't have it set up to simplify `microk8s kubectl` to `kubectl`.

Kubernetes utilizes files created in the YAML format to receive instructions. Does that sound familiar? In *Chapter 15, Automating Server Configuration with Ansible*, we worked with YAML files as that's the format that Ansible playbooks are written in. YAML isn't specific to Ansible; it's used with many different applications and services, and Kubernetes will also recognize the YAML format to contain instructions for how to deploy something. Here's an example YAML file to get us started. This one in particular is intended to launch an NGINX container within our cluster:

```

apiVersion: v1
kind: Pod
metadata:

```

```
name: nginx-example
labels:
  app: nginx
spec:
  containers:
  - name: nginx
    image: linuxserver/nginx
    ports:
    - containerPort: 80
      name: "nginx-http"
```

Before I show you how to run it, let's walk through the file and understand what's going on.

```
apiVersion: v1
kind: Pod
```

First, we're identifying the API version we intend to use, and then we're setting the kind of deployment we intend to set up. In this case, we're deploying a pod, which is what our containers run inside of in Kubernetes. One or more containers can run in a Pod, and a worker node can run one or more Pods at the same time.

```
metadata:
  name: nginx-example
  labels:
    app: nginx
```

Next, we add some metadata. Metadata allows us to set special parameters specific to our deployment. The first item of metadata we're customizing is the name, we're naming the Pod `nginx-example` in this case. We're also able to set up labels with metadata, which is a "name: value" key pair that allows us to add additional values to our Pod that we can refer to later. In this case, we're creating a label called `app` and setting it to `nginx`. This name is arbitrary; we could call it `potato` if we wanted to. Setting this to `nginx` is a descriptive label that makes it obvious to someone else what we are intending to run here.

```
spec:
  containers:
  - name: nginx      image: linuxserver/nginx
```

Moving on, the `spec` section allows us to specify what exactly we want to run in our Pod, and how we want it to run. We want to run a container in our Pod, and specifically a container that we'll name `nginx`, which we'll retrieve from a registry called `linuxserver`, and we're requesting a container from that registry by the name of `nginx`.

The registry we're fetching the container image from deserves some extra explanation. This registry in particular is located at <https://linuxserver.io>, which is a special service that makes container images available for us to download and use. The site has a documentation section that gives us information about each of the container images they offer. Why use the `linuxserver.io` registry? The reason is that their service makes available various container images that support a variety of architectures, including x86 as well as ARM. The latter is especially important, because if you're using Raspberry Pi units for your cluster, they won't be able to utilize container images created for x86. If you do attempt to run a container image that does not support ARM, then the container will fail to launch on a Pi. Since `linuxserver.io` makes container images available for multiple architectures, they should work fine regardless of the type of device you decided to use for your cluster. Whether you're using an x86 physical server or VM for your worker node, the `nginx` container we're retrieving from that registry should function just fine.

```
ports:
  - containerPort: 80
    name: "nginx-http"
```

In the last few lines, we're setting up a container port of `80`, which is the standard for a web server. This is the default port that NGINX listens on when it runs, and NGINX is what we're intending to run inside our container. We're applying a name to this port declaration, and calling it `nginx-http`. We can refer to that name in a subsequent YAML file (if we have more than one) and that works better than having to type the same port in each file. Referring to a port by name is not all that different from variables in scripting or programming languages.

Before we continue, there's a bit of difference regarding how you deploy resources to a cluster within MicroK8s compared to a cluster that was set up manually on actual servers. The commands going forward will assume you've set up a cluster manually. If you're using MicroK8s on Windows or macOS, you'll need to copy any deployment files you create into the MicroK8s VM that's created as part of the installation of MicroK8s. If you try to save a file locally and deploy it as we're about to do in the next paragraph, it will fail, because the file will not be found on the VM. To copy a deployment file to the MicroK8s VM to enable you to deploy it, you can use the following command to copy the file into the VM first:

```
multipass transfer <filename.yml> microk8s-vm:
```

As we go along, keep in mind that whenever we're deploying a file, you'll have to make sure to transfer it first. Also, a manually created cluster uses the `kubectl` command, while MicroK8s requires you to prefix `kubectl` with `microk8s`, so such commands become `microk8s kubectl` instead of just `kubectl`.

Let's go ahead and deploy the container to our cluster. Assuming you named the YAML file as `pod.yml`, you can deploy it with the following command on the controller node:

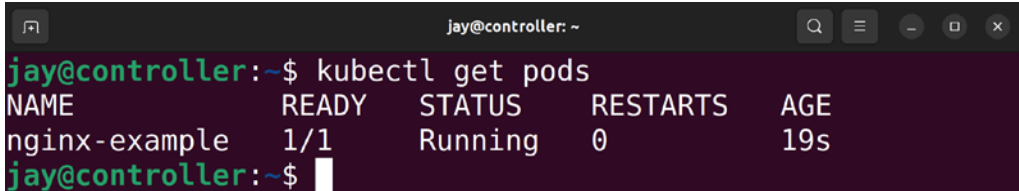
```
kubectl apply -f pod.yml
```

As mentioned previously, the `kubectl` command allows us to control our cluster. The `-f` option accepts a file as input, and we're pointing it to the YAML file that we've created.

Once you've run the command, you'll be able to check the status of the deployment with the following command:

```
kubectl get pods
```

For me, I see the following when I run it:

A terminal window with a dark purple background. The prompt is 'jay@controller: ~'. The command 'kubectl get pods' has been entered and executed. The output is a table with five columns: NAME, READY, STATUS, RESTARTS, and AGE. The first row shows 'nginx-example' with '1/1' in the READY column, 'Running' in the STATUS column, '0' in the RESTARTS column, and '19s' in the AGE column. The prompt 'jay@controller: ~\$' is visible at the bottom.

NAME	READY	STATUS	RESTARTS	AGE
nginx-example	1/1	Running	0	19s

Figure 18.20: Checking the status of a container deployment

From the output in my case, I can see that the process was successful. The STATUS is Running. What we don't see is what worker node the Pod is running on. It's nice to know that it's Running, but where? We can get more information by adding the `-o wide` option to the end of the command:

```
kubectl get pods -o wide
```

The output contains much more information, so much that a screenshot of it won't fit on this page. The focus though, is that among the extra fields we have with this version of the command is the node field, which shows us which worker node is handling this deployment. We'll even see an IP address assigned to the Pod as well.

We can use the IP address shown for the Pod to access the application running inside the container. In my case, my Pod was given an IP address of `10.244.1.2`, so I can use the `curl` command to access the default NGINX web page:

```
curl 10.244.1.2
```

The output that's shown in the terminal after running that command should be the HTML for the default NGINX web page. Note that the `curl` command may not be available in your Ubuntu installation, so you may need to install the required package first:

```
sudo apt install curl
```

We have one potential issue though: if you want to be able to access an application running inside the cluster on a machine other than the controller or worker nodes, it won't work. By default, there's nothing in place to route traffic from your LAN to your cluster. This means that in my case, the IP address of `10.244.1.2` that was given to my Pod was provided to it by the Pod Network, the router on my network doesn't understand that IP address scheme so trying to access it from another machine on the LAN will fail. What's interesting is that you can access the application from any other node within the cluster. In my case, the NGINX Pod is running on the first worker node. However, I can actually run the previous `curl` command from worker #2 even though the Pod isn't running there, and I'll get the same exact output. This works because the Pod Network is cluster-wide; it's not specific to any one node.

The IP address of `10.244.1.2` is unique to the entire cluster, so if I run another container it won't receive that same IP address, and every node within the cluster knows how to internally route to IPs within that network.

Not allowing outside devices to access applications within the cluster is great for security purposes. After all, a hacker can't break into a container that they can't even route to. But the entire point of running a Kubernetes cluster is to make applications available to our network, so how do we do that? This can be a very confusing topic for newcomers, especially when we're setting up a cluster manually. If we're using a cloud service such as AWS or Google Cloud, they add an additional layer on top of their Kubernetes implementation that facilitates routing traffic in and out of the cluster. Since we set up our cluster manually, we don't have anything like that in place.

When it comes to building services and networking components designed to handle networking to bridge our LAN and Kubernetes cluster, that's an expansive topic that can span a few chapters in and of itself. But a simple solution for us is to create a **NodePort Service**. These are two new concepts here, a Service as well as NodePort. When it comes to a Service, a Pod isn't the only thing we can deploy within our cluster. There are several different things we can deploy, and a Service is a method of exposing a Kubernetes Pod, and a NodePort is a specific type of service that gives us a specific method for facilitating a way of accessing it. What NodePort itself does is expose a port running inside a Pod to a port on each cluster node.

Here's a file we can deploy to create a NodePort Service for our Pod:

```
apiVersion: v1
kind: Service
metadata:
  name: nginx-example
spec:
  type: NodePort
  ports:
    - name: http
      port: 80
      nodePort: 30080
      targetPort: nginx-http
  selector:
    app: nginx
```

As you can see, the `kind` is `Service`; as this time, we're deploying a service to our cluster to complement the Pod we've deployed previously. The type of service is `NodePort`, and we are mapping port `80` within the Pod to port `30080` in our cluster. I chose port `30080` arbitrarily. With `NodePort`, we can utilize ports `30000` through `32767`. The `selector` in this file is set to `nginx`, which is the same selector we used while creating the Pod. Selectors allow us to "select" Kubernetes resources via a name we assign them, to make it easier to refer to them later.

Let's deploy our service:

```
kubectl apply -f service-nodeport.yml
```

Assuming we've typed everything in the YAML file for the service properly, we can retrieve the status of services running within our cluster with the following command:

```
kubectl get service
```

If everything has gone well, you should see output similar to the following:



```

jay@controller:~$ kubectl get service
NAME                TYPE          CLUSTER-IP    EXTERNAL-IP    PORT(S)          AGE
kubernetes          ClusterIP     10.96.0.1     <none>         443/TCP         85m
nginx-example       NodePort      10.108.225.118 <none>         80:30080/TCP    10s

```

Figure 18.21: Checking the status of a service deployment

In the output, we should see the status of our service deployment, which is the second line in the screenshot. We can also see the port mapping for it, showing that port 30080 should be exposed to the outside. To test that it's actually working, we can open a web browser on a machine that is within our LAN, and it should be able to access the controller's IP address at port 30080:

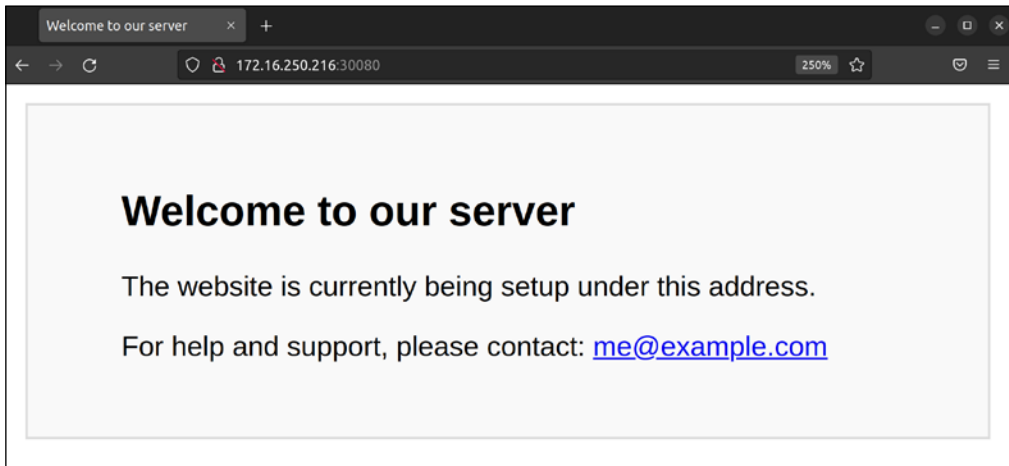


Figure 18.22: Accessing a web page served by an NGINX container in a cluster from within a web browser

Another interesting benefit is that we didn't actually have to use the IP address of our controller node, we can use the IP address of a worker node as well and see the same default page. The reason this works is that the mapping of port 30080 to port 80 inside the Pod is cluster-wide, just as the internal Pod Network is also cluster-wide. Accessing a resource on one is the same as accessing the same resource on any other, as the request will be directed toward whichever node is running the Pod that matches the request.

When it comes to removing a Pod or a Service, assuming you want to decommission something running in your cluster, the syntax for that is fairly straightforward. To remove our `nginx-example` Pod, for example, you can run this:

```
kubectl delete pod nginx-example
```

Similarly, to delete our service, we can run this:

```
kubectl delete service nginx-example
```

At this point, you not only have a working cluster, but you also have the ability to deploy containers to it and set them up to be accessible from outside the Pod Network. From here, I recommend you practice with this a bit and attempt to run additional containers and apps to have a bit of fun with it.

Summary

In this chapter, we took containerization to the next level and implemented Kubernetes. Kubernetes provides us with orchestration for our containers, enabling us to more intelligently manage our running containers and implement services. Kubernetes itself is a very expansive topic, and there are many additional features and benefits we can explore. But for the goal of getting set up on Kubernetes and running containers with it, we did what we needed to do. I recommend that you continue studying Kubernetes and expand your knowledge, as it's a very worthwhile subject to dig deeper into.

Speaking of subjects that are worthwhile to learn, in the next chapter, we're going to learn how to deploy Ubuntu in the cloud! Specifically, we'll get started with Amazon Web Services, which is a very popular cloud platform.

Relevant videos

- Setting up a Kubernetes Cluster (LearnLinuxTV): <https://linux.video/setup-k8s>

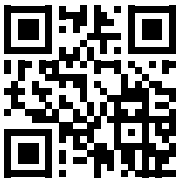
Further reading

- MicroK8s website: <https://learnlinux.link/mk8s>
- MicroK8s addons: <https://learnlinux.link/mk8s-addons>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaz0>



19

Deploying Ubuntu in the Cloud

Up until now, in each chapter, we've been working with an instance of Ubuntu installed on either a local virtual machine, a physical computer or server, or even a Raspberry Pi. We've learned how to deploy Ubuntu on such devices, and we've even gone as far as deploying virtual machines as well as containers. These on-premises devices have served us well, but the concept of cloud computing has become quite popular, even more so since the previous edition of this book. In this chapter, we're going to take a look at running Ubuntu in the cloud. Specifically, we'll deploy an Ubuntu instance on **Amazon Web Services (AWS)**, which is a very popular platform for cloud computing. While we won't go into extreme detail on AWS (it's an extremely large and complex platform), you'll definitely get a feel for what it's like to deploy resources in the cloud, which will be more than enough to get you started.

This exploration will involve the following topics:

- Understanding the difference between on-premises and cloud infrastructure
- Important considerations when considering cloud computing as a potential solution
- Becoming familiar with some basic AWS concepts
- Creating an AWS account
- Choosing a region
- Deploying Ubuntu as an AWS EC2 instance
- Creating and deploying Ubuntu AMIs
- Automatically scaling Ubuntu EC2 deployments with Auto Scaling
- Keeping costs down: understanding how to save money and make cost-effective decisions
- Taking the cloud further: additional resources to grow your knowledge

Since cloud computing is something of a different mindset than we're used to, we'll first go through a few sections that are dedicated to helping us understand the difference, as well as some of the considerations we should make before choosing to implement Ubuntu in the cloud. In the next section, we'll explore how the concept of cloud computing differs from on-premises hardware.

Understanding the difference between on-premises and cloud infrastructure

As mentioned at the very beginning of this chapter, we've been solely utilizing on-premises Ubuntu installations thus far. Even if we're running Ubuntu on a virtual machine in our data center, it's still considered an on-premises installation even when it's not on physical hardware. In short, an on-premises installation is something that resides locally with us, regardless of the type of server that serves as the foundation.

The first difference when it comes to cloud computing might be somewhat obvious: it's the exact opposite of a resource being on-premises. With a cloud instance of Ubuntu, it's someone else's hardware that it runs on. Most of the time, we won't know what kind of server a cloud instance is running on—when we subscribe to the services of a cloud provider and pay a fee to run a server on that platform, we're able to access the operating system just like we would a virtual machine, with little knowledge of the underlying data center. Although utilizing a cloud instance will have some sort of recurring cost, we don't need to worry about monitoring hardware components or replacing failed physical devices. With the cloud, that becomes someone else's problem.

There are various cloud providers you can choose from, with **AWS**, **Google Cloud Platform (GCP)**, and **Microsoft Azure** being popular choices. Those are three very popular platforms. Each of them has its own strengths and weaknesses, and even some differentiation as far as how resources on the platform are managed. For the most part, there's no right or wrong choice here, it's important to research each of the platforms and make the appropriate decision for you and your organization.

In addition to cloud providers, there are also **Virtual Private Server (VPS)** providers. There used to be more of a distinction between a full cloud provider and a VPS provider – but as time goes on, the line between the two types of platforms is becoming increasingly thin. For the most part, VPS providers are lighter or “simpler” alternatives to standard cloud providers. They provide the same core features, such as allowing you to build virtual machines – but they don't usually offer as many features as full cloud providers do. Nowadays, VPS providers are rolling out new features regularly and are catching up to cloud providers in terms of features. Examples of popular VPS providers include **DigitalOcean** and **Linode**, among others. In fact, even Amazon themselves have gotten into the VPS business, offering a VPS alternative to their own platform, called **Amazon Lightsail**.

For your organization, which should you choose between a full cloud provider and VPS provider? Many administrators would answer this question without hesitation and recommend a full cloud provider. I can certainly see their point – why go with a “lighter” platform when you can use a solution that has all the bells and whistles? However, based on my experience with many organizations, I wouldn't recommend going with a full cloud provider without first finding out if a VPS provider has all the features you need. The reason for this is that one of the trade-offs of having more features is that complexity increases. With an increase in complexity comes a larger responsibility to manage resources on that platform. And that complexity can create a very large administration overhead. For that reason, I don't recommend full cloud providers like AWS or Azure to small organizations with limited IT staff. For those organizations, VPS providers would be best as the burden of administration is lower. Larger organizations with a team of administrators should have no problem tackling something like AWS.

As I mentioned, we'll be going over AWS in this chapter. Even if you're not considering migrating resources to the cloud, it's something that's still worth practicing. If nothing else, you'll gain some familiarity with another platform, which will increase your skillset and marketability. And I don't know about you, but speaking for myself, I really enjoy checking out technologies and learning something new.

However, perhaps I'm getting ahead of myself. Before you migrate anything to the cloud, the most important question to ask right now is whether or not creating cloud resources is appropriate for your goals. That's exactly what we'll explore in the next section. We'll also explore some of the differences in mindset when it comes to cloud computing as well.

Important considerations when considering cloud computing as a potential solution

Before choosing to sign up with a provider, it's important to first make sure that creating cloud resources is a good idea for you or your organization in the first place. Often, IT professionals can get so excited when it comes to a new trend that they may make the mistake of trying to use such a service even when it doesn't make sense to do so. Above all, as an administrator, it's important to utilize the best tool available for whatever it is that you wish to accomplish, instead of using a technology just because you're excited about it. Cloud computing is awesome for sure, but for some use cases, it's just not a good fit. This is similar to containers as well: containerization is an exciting technology but some applications just don't run well on that platform. It takes trial and error.

There are some considerable benefits when it comes to cloud computing. When it comes to physical servers, the hardware will fail eventually. It's not a matter of "if" but "when." All hardware *will* fail eventually. And even if you have hardware that doesn't end up failing in the short term and lasts for a long time, it will be made obsolete by more powerful hardware that is more efficient. When it comes to managing physical servers, you'll need to replace the hardware eventually.

This is also true when it comes to cloud computing; the hardware such services run on will need to be replaced whenever it fails or becomes obsolete (whichever happens first). The difference is that when that happens, the liability on you (the administrator) is greatly reduced. You won't have to order a new server, replace parts, or even pay attention to the hardware at all. It's solely the responsibility of the cloud service or VPS provider to keep track of that. A virtual machine you run on a cloud platform could be running on a physical server that's a couple of years old, and then tomorrow be moved to a brand-new piece of hardware. And you might not even be aware that it was ever moved.

The trade-off, though, is potentially the cost. The reason why I mention cost as a "potential" trade-off is that whether or not it's cheaper for your organization to purchase physical servers or pay a monthly fee for servers in the cloud comes down to which offers a better **Return on Investment (ROI)**. To better understand this, consider an organization that utilizes only on-premises hardware for their data center with no cloud resources being used at all. Let's also assume that every 3 years or so, they have to replace some of the more critical servers with newer hardware; and every 5-10 years the less-important servers are replaced. There's also a need to pay for full-time administrators with specialties in managing physical hardware, as well as maintaining a cooling system and making sure a room is available to designate as a data center. Also, consider the electric bill for running your own data center. Add all of this up—how much is it costing the organization on average?

With physical infrastructure, you're paying an up-front cost to purchase servers every time you need to do so. With cloud computing, you never have to purchase physical server hardware, but instead you're paying a fee each and every month for the privilege of utilizing cloud infrastructure. When you add up the monthly costs for cloud resources, how much would that cost the organization? Would it cost more than running physical hardware or less?

Often, whether or not running a server in the cloud results in cost savings (or ends up being cost prohibitive) comes down to what types of services you wish to use on that platform. Running more virtual machines in the cloud is generally fairly inexpensive, and might often result in cost savings when compared to running physical infrastructure of your own. However, an example of something that might end up being cost prohibitive in the cloud is storage.

If your organization doesn't store much data, then storage costs won't really be a concern. Many companies are perfectly satisfied with employees utilizing something like Google Drive for shared storage and won't even need cloud storage at all. Some organizations, especially those that develop software, have *massive* storage needs. For these companies, there could be tens or even hundreds of terabytes of data being stored, as well as the bandwidth costs for the data going in and out of the company. If you were to attempt to migrate such a large amount of data to a cloud server provider, you wouldn't have to manage the storage hardware anymore, but your costs for cloud storage would go up dramatically and might easily be much more expensive than it costs you to continue to use physical hardware. Personally, I've seen storage costs result in tens of thousands of dollars extra in monthly billing.

Another consideration is stability. It's often argued that utilizing a cloud provider will result in a more stable infrastructure. That mindset seems to make sense at first, considering that by using a cloud provider you're no longer responsible for managing the hardware. There's definitely some truth to the claim that cloud resources are more stable, but it's not quite that simple. Cloud providers *do* experience downtime, and it happens more often than you'd think.

Cloud providers will often advertise a high level of uptime, often measured in "9s." For example, AWS offers a general uptime of 99.999%, as of the date this book is being prepared for publishing. That sounds great, doesn't it? Before you get too excited about a cloud provider's uptime claim, it's important to also know what exactly they count as part of that uptime. If an upstream provider, such as a backend internet resource, goes down and service is completely unavailable, they may not count that and continue to claim the same amount of uptime. Massive outages of cloud providers are not uncommon; in 2017, Amazon's S3 service suffered a major outage due to one of their engineers mistyping a command. In that situation, many services on the internet were unavailable to customers. So while it is true that a cloud service provider is less likely to suffer an outage than you are to witness hardware failures, it's important to keep in mind that outages are still possible. We can't ever assume any service (our own or otherwise) is bulletproof.

Automation is a very important consideration with the cloud. If your resources encounter an issue, it's a really good idea to have an automated means of re-deploying your important services. On a physical server, you can set it up exactly as you want it and take an image of the hard drive in case the server fails. Cloud providers offer you essentially the same service, giving you the ability to create images of your important servers so you can redeploy them in the future should you need to do so.

If you do go with a cloud provider, make sure you keep regular backups as well as images. And be sure to keep a copy of at least the most recent backups locally outside of the cloud provider, because if the cloud provider itself goes down then you also lose the backup. I can probably better summarize this by advising you to not become over-confident in the stability of your cloud provider. Always assume your infrastructure will fail eventually, regardless of where it's located or how stable their marketing team claims it is.

The warnings I have given around the stability of cloud providers are not intended to scare you away from using them, but instead to steer you toward maintaining good hygiene with the cloud, just as you would with physical infrastructure. Basically, to be successful when it comes to managing servers, you'll have to assume that cloud providers can fail just as easily as on-premises equipment, and make sure you're well prepared if there is ever an issue. Cloud providers actually provide you with all the tools you need in order to build a stable infrastructure that's easily reproducible and recoverable. If you utilize those tools effectively, you shouldn't have anything to worry about. We'll explore these concepts further as we go along, but you can go as far as to have cloud servers recover *themselves* when a problem occurs. It all depends on how you architect your solution.



It's also important that you understand any company policies (and laws) that may exist regarding data retention, what type of information is allowed to be stored within the cloud, or any other rule or regulation that might be in place in your area. (GDPR is a great example of legal requirements.) If in doubt, consulting with your company's management, security, and human resources teams would be a great place to start when it comes to what data-related policies and laws may be applicable while you design your cloud solution.

In the next section, we're going to explore some concepts specific to AWS, so we'll have a stronger foundation of knowledge to use for building an actual cloud solution later on in this chapter.

Becoming familiar with some basic AWS concepts

As discussed earlier, AWS is one of several competing cloud service providers. For the purpose of this chapter, AWS was chosen because more than any other provider, the platform requires an administrator to adopt a completely different mindset when it comes to managing infrastructure. This different mindset is a healthy one even outside of AWS, so it represents a logical evolution at this point in our journey.

Up until now, we've discussed server installations as essentially pets, meaning we want to keep them around, make sure they're healthy, and if something goes wrong, try to fix it. We want to keep our servers operational for as long as possible. We want to be able to rely on them, and that helps our organization - customers and clients appreciate using a website or service that is stable, with minimal or no downtime.

That last part, minimal downtime, doesn't change regardless of the mindset we use when managing our infrastructure. Downtime and service disruption is bad in this industry, and that's always going to be the case. The difference is what we do about it, and how we go about recovering from it. In fact, we can even try our best to automatically recover when we have a problem. If the customer never notices there was ever an issue, even better.

With AWS, the mindset is different – we consider our servers as disposable, not as pets. This may seem a bit surprising at first, but if we effectively use the tools we’re provided, we can build an infrastructure that is scalable. This concept in particular is known as **Auto Scaling** and is a very important aspect of AWS. With Auto Scaling, resources are automatically created and destroyed as demand increases and decreases. For example, let’s say that your web server receives more visitors than normal, and its CPU is starting to max out. Auto Scaling would then bring up a new web server automatically, and with a load balancer, route clients between instances as needed to spread the load. You can set the maximum number of instances that can be brought online, and then when the load decreases, the servers that were brought online to handle the load will automatically get deleted. This means that you can architect your cloud solution such that you’ll have the exact number of servers in existence as you need at any given time.

Another level above Auto Scaling is **Auto healing**. Just as the name implies, auto healing means that if your server runs into a problem, it will automatically be “healed,” and in the case of cloud computing, this means the instance will be disposed of and recreated from a known good image or template. This is the ultimate goal of AWS (or pretty much any implementation): to have infrastructure that is not only scalable but also able to recover by itself. For example, perhaps you have a pair of web servers that handle requests from clients. One of them encounters some sort of issue and fails some sort of test that would imply that the server is unreliable. Such a test is known as a **health check**. With auto healing, a server that fails health checks is considered unhealthy and is deleted. With Auto Scaling, you also set a minimum number of servers for your application. If the number of servers falls below the minimum, then a new one is created to replace it. Depending on how you architect your solution, the customer may notice some degradation if the application is running on fewer servers, but that’s only temporary. Everything will return back to normal when the new server comes online.

With AWS, Auto Scaling and auto healing are extremely important. I’ve seen many administrators make the mistake of not utilizing those features of the platform, and that should never be the case. Without such high-availability features, you risk having your server go away at any moment. This may be a bigger problem than you think. AWS consists of physical servers all over the world, and just like any other physical server, hardware failures are not only possible but happen regularly. If your application is running on a physical host within an AWS data center that encounters a hardware failure, your server may get deleted when they go and replace it. With Auto Scaling, this isn’t a problem. A new virtual server will be brought online to replace the one that was removed.

But without Auto Scaling, this can result in manually having to rebuild your server. Don’t worry though, we’ll cover Auto Scaling in this very chapter, so you’ll definitely understand how to implement it before we’re done.

The concepts around high availability are not limited to AWS. Similar features exist on other cloud platforms as well. The main difference between competing cloud providers is the marketing terms they use for these features, but you can set up similar infrastructure on each. With AWS, they really focus on this aspect though, so it’s important to learn. But even if you’re not utilizing AWS in production at your organization, the concepts around being able to easily recover from disasters are still important and can still be implemented. If nothing else, you should at least consider implementing automation for rebuilding servers so you don’t have to do so much manual work if a problem occurs.

Back to AWS: there are many services within the platform that you can use that will provide a variety of features. In fact, there are well over a hundred different services within the platform. Therefore, it's impossible for us to cover each in this chapter. It would be difficult to cover every service even in a book dedicated only to AWS. But don't let the number of services overwhelm you; you'll only need to learn about the services that are related to what you're trying to achieve with the platform. For example, if your organization doesn't develop multiplayer computer games, then learning the GameLift service will be of no benefit to you at all. In this chapter, we'll focus on the services required to get you up and running with a basic web server running in the cloud. The following services and terms will be discussed:

- **Virtual Private Cloud (VPC):** A VPC is a higher-level abstraction of your overall cloud network and resources. Each of the servers and related services you provision will fit within a VPC. You can think of a VPC as your overall network. In your organization, you may have various routers, gateways, firewalls, virtual machines, printers, or other network-connected devices. Your organization's network is essentially a VPC in AWS, a software version of a complete network.
- **Elastic Compute Cloud (EC2):** EC2 is the service within AWS that your virtual machines will run in. Individual virtual machines within AWS are referred to as EC2 instances. You can think of this service as the AWS equivalent of VMware ESXi, Microsoft Hyper-V, Proxmox, or whatever virtual machine platform you're more familiar with. EC2 instances, just like a virtual machine, have memory and CPU allocated to them and run an operating system. We'll run Ubuntu in an EC2 instance before the end of the chapter.
- **Elastic Block Store (EBS):** As you learn more about AWS, you'll notice that even the simplest component seems to have a marketing buzzword attached. EBS provides block storage, essentially the same type of storage we've been working with all along. So in a nutshell, an EBS volume is a hard disk. When you create an EC2 instance, the operating system for your server will run from an EBS volume, and you can set the size of the volume accordingly. We'll work through this later in the chapter.
- **Elastic Load Balancer (ELB):** As you may be able to guess from the name, an ELB is the AWS equivalent of a load balancer and offers similar features. This allows you to have multiple EC2 instances serving your application, and you can create an ELB to route traffic between them. ELB is actually a feature of EC2 and not its own service.
- **Identity and Access Management (IAM):** IAM is the tool within AWS that you'll use to create and manage user accounts, determine user permissions, and even create API keys that can be used to programmatically access and manage AWS. Basically, it's your one-stop shop for all things related to user privileges, regardless of whether the "user" is a human or a script.
- **Route 53:** Although we're not going to cover Route 53 in this book, I recommend at least understanding what it is in case you need it in the future. If you do decide to utilize AWS in production, Route 53 will simplify the process of managing DNS entries and also registering new domain names. If your organization is a managed service provider, you may find yourself using this quite a bit.
- **Simple Storage Service (S3):** Amazon's S3 service is another offering we're not going to cover in this chapter, but it's a good idea to know that it exists and what it's for, in case you find a use for it later. S3 is actually a very popular service, and it provides **object storage**.

Object storage is a new type of storage that is different than a disk (virtual or physical) that you add to your server, format, and mount. While you can still mount S3 on your server, it doesn't have a filesystem (such as ext4 or STON), nor does it understand permissions. It's simply a name-object pair, where you store files, and they have a name. With S3, you create "buckets," and each bucket can have files stored inside. Each bucket name must be unique. S3 is very useful if you want to make downloadable files available to your clients or store backup files.

- **Elastic Kubernetes Service (EKS):** In the previous chapter, we covered Kubernetes and even set up our own cluster. AWS has its own Kubernetes solution, called EKS. Although we're not going to cover it in this book, it's worth considering if you want to continue to use Kubernetes and have your containers running in a managed service, rather than managing your own cluster. EKS combines Kubernetes with the flexibility of the AWS platform, so it's a very useful service to consider.
- **Security groups:** Access to many AWS resources from the public internet is disabled by default; security groups are used to determine what is able to access a resource within AWS, and you can allow or disallow access by IP address as well as port. With regard to EC2 instances, outbound access is allowed by default, but every port is blocked inbound. You can create a security group that allows specific IPs to access the instance, which increases security. We'll see an example of this later.

Now that we have some basic understanding out of the way, we can get started and build an application in the cloud, running on AWS.

Creating an AWS account

As mentioned in the previous section, a VPC within AWS represents a high-level abstraction of your overall network. All of the resources that we create will run inside a VPC. Therefore, we'll need to create a VPC first before we can create an EC2 instance and deploy Ubuntu.

Before we can create a VPC though, we'll need an AWS account. Before this chapter, I typically advised you to use whatever hardware you have available in order to create Ubuntu installations to work with the platform. This time, we're going to utilize an actual cloud provider, which comes at a cost. While there are free components available for a limited time with a new account, it's up to you, the reader, to keep track of billing. We'll discuss costs in greater detail later in this chapter. But as a general rule of thumb for now, always use whatever the cheapest option is. If a free instance type is available, go with that. Of course, if you're intending to deploy actual resources for production use within an organization, then you'll want to choose whatever instance type is appropriate for the use case. For our purposes, we're just learning the platform for the first time, so be sure to go with the lowest-cost resources available and delete everything when you're done.

That last point is especially important – the free tier is not unlimited, and if you forget to delete something, you might receive a bill at some point. What I recommend is to keep a running list of everything you create in AWS, so you have a list of things to delete if you don't wish to continue using the platform.

You should also remove any snapshots/storage volumes, or anything else that you end up creating. Better yet, if you aren't going to use the AWS account in production, you can simply delete the account so that it doesn't continue to bill you. If ever in doubt, be sure to consult the documentation for billing within AWS itself.

Signing up for AWS

Now it's time for us to create our own AWS account. In this section (and in each of the remaining sections of this chapter) I'll walk you through each step when it comes to navigating AWS. It's important to keep in mind that the **AWS Management Console** that we'll be navigating has a history of changing quite often. In fact, the console has changed no fewer than twice since the previous edition of this book was written, and that's not even counting the fact that in the third edition, the interface changed in the middle of writing the previous version of this chapter. The screenshots and walkthrough are accurate as of the time this chapter has been written, but it's always possible (and quite likely) that the interface will be updated *again* at some point. That said, if the screenshots or layout have changed, you should still be able to follow the instructions here since the verbiage generally doesn't typically change, just the layout. With that said, let's get started.

To begin, navigate to `https://aws.amazon.com` and you should see an orange button in the upper-right corner with the label **Create an AWS Account**. It's possible that the layout of the page may change after publication, but you should see a button to create a new account somewhere on the page:

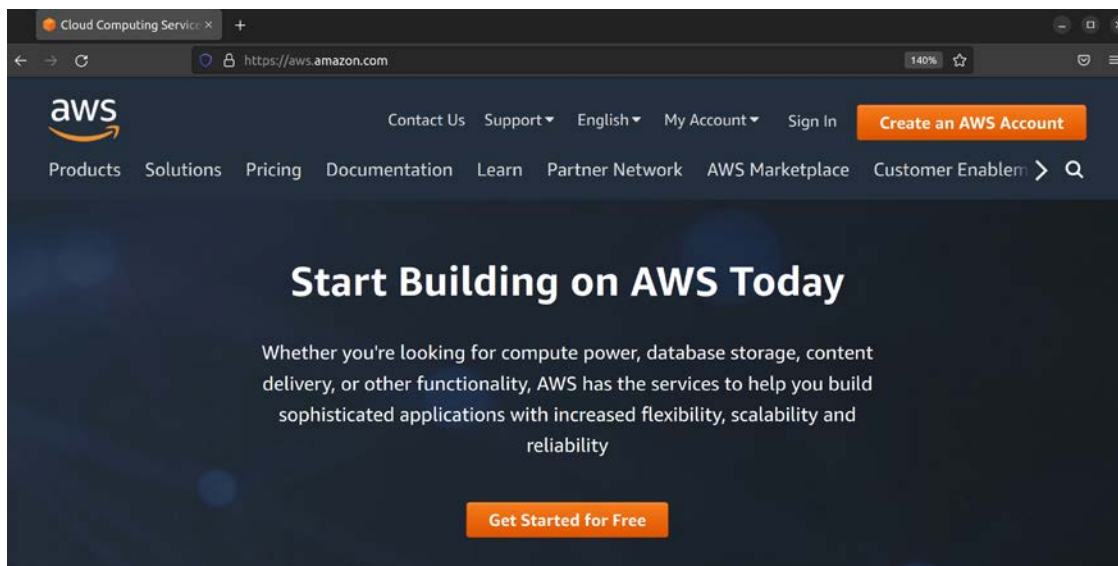
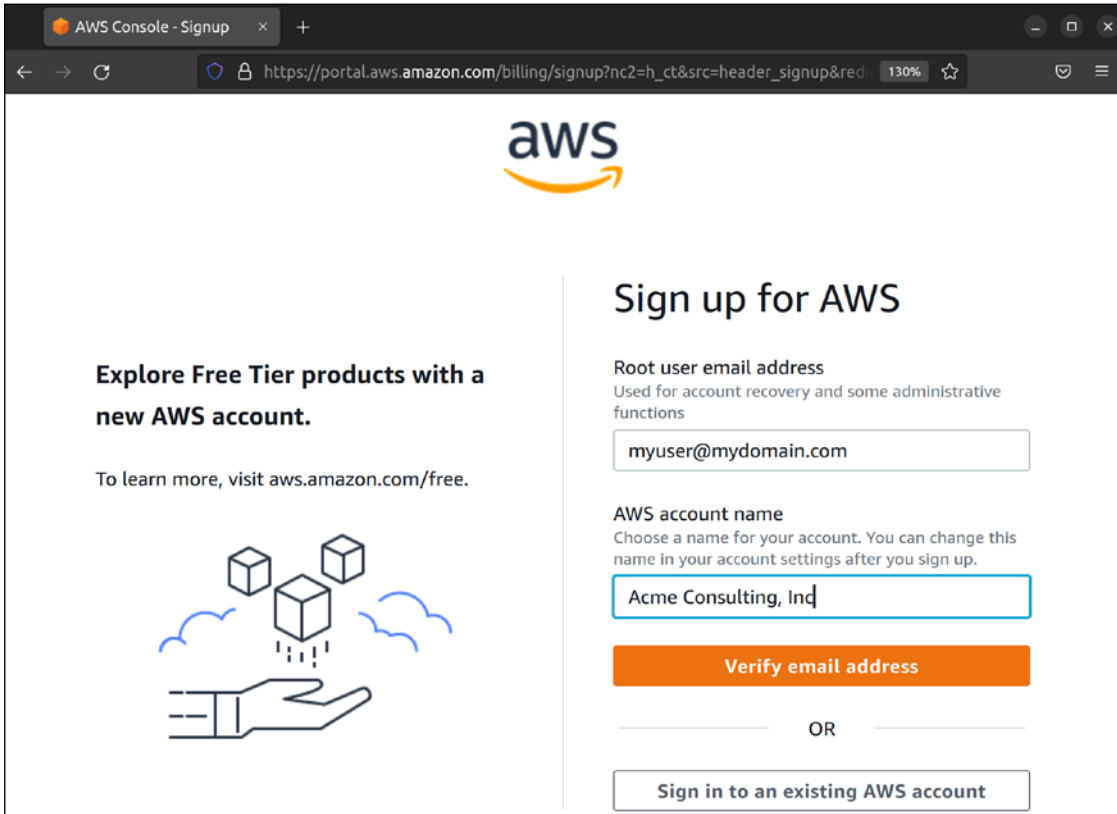


Figure 19.1: The AWS main page

When you click the button to create a new account, a form will appear asking you for basic information. An example of some of the fields you may be asked for is shown in *Figure 19.2*, though the exact information you may be asked for may vary. Fill out each field accordingly, then click **Continue**:



The screenshot shows a web browser window with the title "AWS Console - Signup" and the URL "https://portal.aws.amazon.com/billing/signup?nc2=h_ct&src=header_signup&red". The page features the AWS logo at the top center. On the left side, there is a section titled "Explore Free Tier products with a new AWS account." with a sub-link "To learn more, visit aws.amazon.com/free". Below this is an illustration of a hand holding three server racks. On the right side, the main heading is "Sign up for AWS". The form includes the following fields and buttons:

- Root user email address**: Used for account recovery and some administrative functions. The input field contains "myuser@mydomain.com".
- AWS account name**: Choose a name for your account. You can change this name in your account settings after you sign up. The input field contains "Acme Consulting, Inc".
- Verify email address**: An orange button.
- OR**: A separator between the two options.
- Sign in to an existing AWS account**: A button at the bottom.

Figure 19.2: Signing up for a new AWS account

After you click **Continue**, another screen will appear asking you to fill out additional fields, such as your full name, company name, address, and so on. This process will also include asking you to provide credit card details as well, so proceed through each screen and enter the required information. At the end of the process, you'll see a selection where you can choose your support plan:

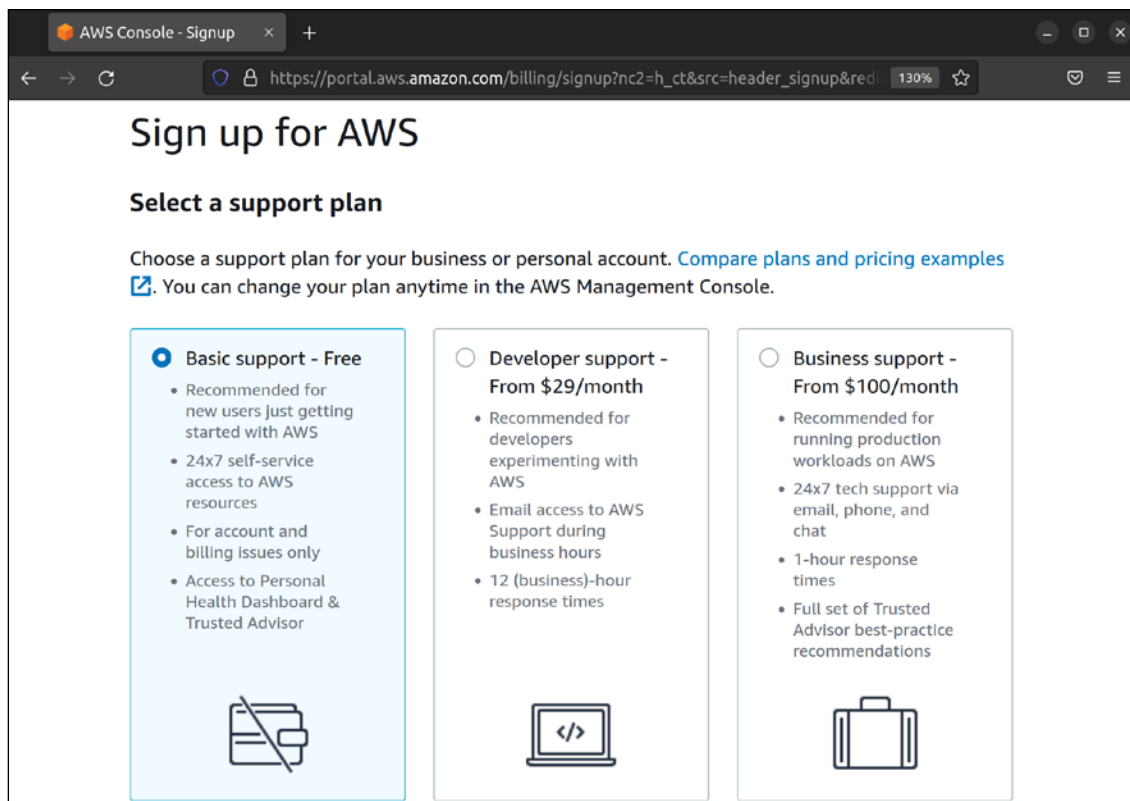


Figure 19.3: Choosing a support plan while signing up for an AWS account

If you are asked to select a support plan during the process, choose the one that makes sense for your use case. If you're going to be using AWS to create production instances for your organization, the Developer or Business plan may offer additional value to you. If you're only going through the process to learn AWS and work through the examples in this chapter, choose the Basic plan.

Once you've finished the process of creating your new account, it may take some time before your new account is ready. When it is, you should receive an e-mail saying that it's ready for use. After you do receive that e-mail, you'll be able to log in. To do so, navigate to `https://aws.amazon.com` and click the **Sign In to the Console** button, which will bring you to another page where you can type in your account login info:

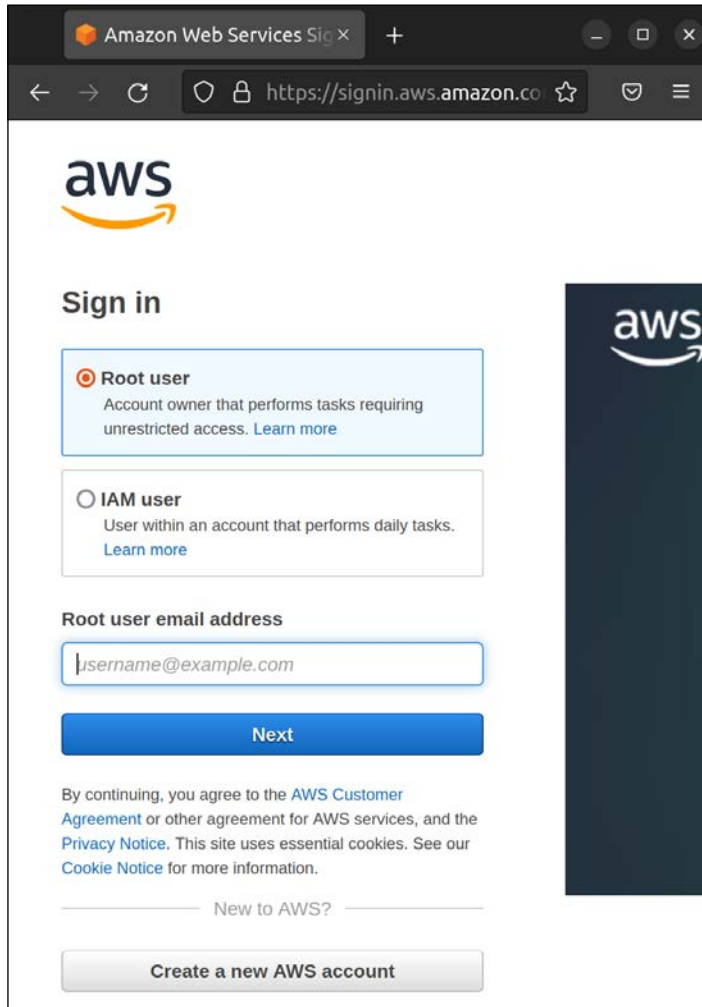


Figure 19.4: Signing in as Root user

Here, you have a choice to log in as the root user or an IAM user. We haven't created any IAM users yet, so we only have a root user at this point. The root user is accessed by signing in with the same e-mail address that you've provided during the sign-up process. Enter that e-mail address in the prompt, click **Next**, then enter your password on the next screen.

If all goes well, you should be logged in and see the **AWS Management Console**:

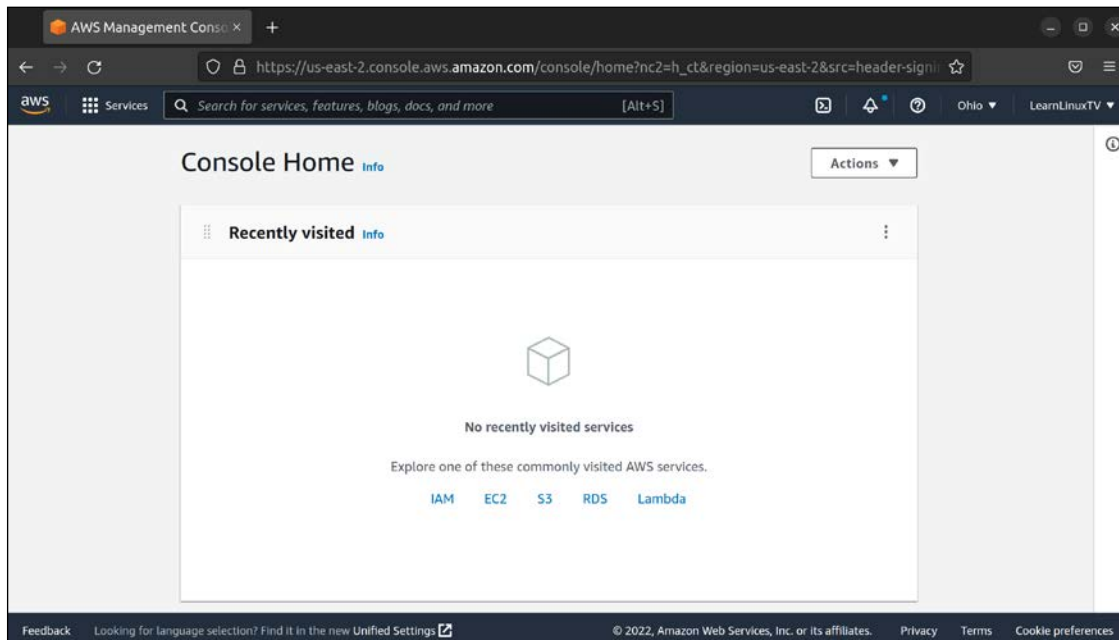


Figure 19.5: The AWS Management Console main screen

Now that you have access to the management console, you can begin using AWS. Before we start creating cloud resources, we should implement some basic security to protect our account.

Implementing basic user security

Before we continue any further, there are some very important security best practices we should employ regarding the ability to authenticate to our AWS account. Although it's very likely that our new account was created as a test account for following along with examples in this chapter, we should make it a habit to always protect our AWS account, regardless of how important it actually is.

We can begin with protecting the root account, as it's a common target for hackers. Specifically, we should enable two-factor authentication for this account. Doing so will make it much harder for an outside threat to access it, since they would need access to your second factor in addition to your password. Since we're already logged in as the root user at this point, we can set this up immediately.

In the management console, you'll see a search field near the top of the screen. If you already know the name of the service you would like to configure, you can begin typing its name in the search field, and if your query matches an available service, it will show it in the list. If you don't know the name of the service you'd like to use, you can click on **Services** in the top-left corner of the console, to see a complete list of the services available.

For setting up second-factor authentication, we will access the IAM service. You can start typing IAM into the search field, and it should show it as an available option. Go ahead and click on it:

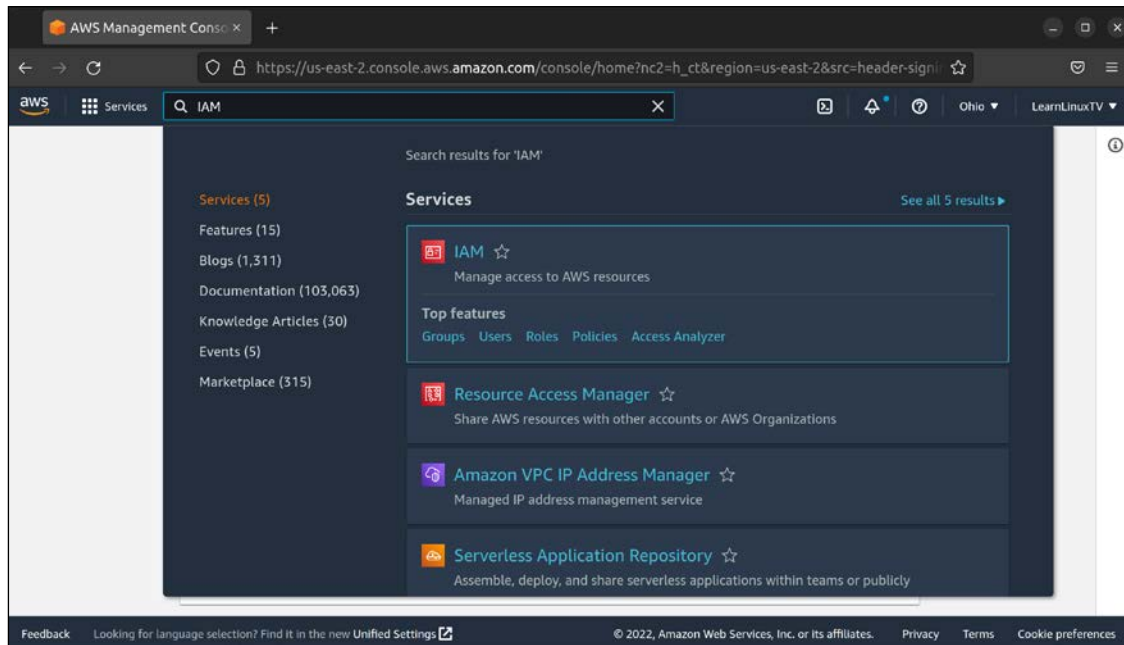


Figure 19.6: Using search within the AWS Management Console to locate the IAM service

When the IAM dashboard appears, you may see a security alert that complains about the root user not having **Multi-Factor Authentication (MFA)** enabled. Although enabling this feature is optional, I recommend you enable it.

MFA is critical for ensuring the security of any account that is important to your organization, especially an account for a cloud computing provider such as AWS. MFA will enhance the security of the account in terms of authentication, requiring an additional factor before a user will be able to access the account. The root account is the most critical account to protect in AWS, since it has full access to every available service. To enable MFA, click on the button that reads **Add MFA** (or similar verbiage if the layout on this page changes):

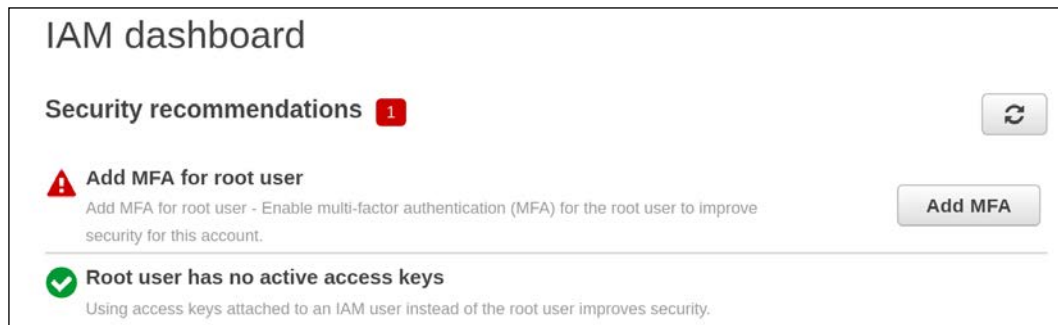


Figure 19.7: IAM dashboard, with an alert that MFA is recommended to be set up

Next, click on **MFA** to expand it (if it isn't already), and then click on the **Activate MFA** button:

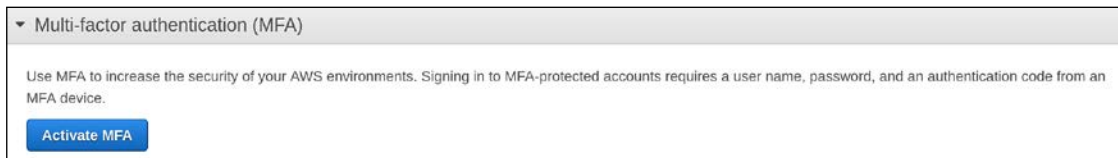


Figure 19.8: Setting up MFA (continued)

The next screen will give you a choice of which type of device to use to facilitate MFA. The default (**Virtual MFA device**) is a good one to go with if you don't have a physical hardware key, such as a YubiKey. If you have no other preference, choose the **Virtual MFA device** option and click **Continue**:

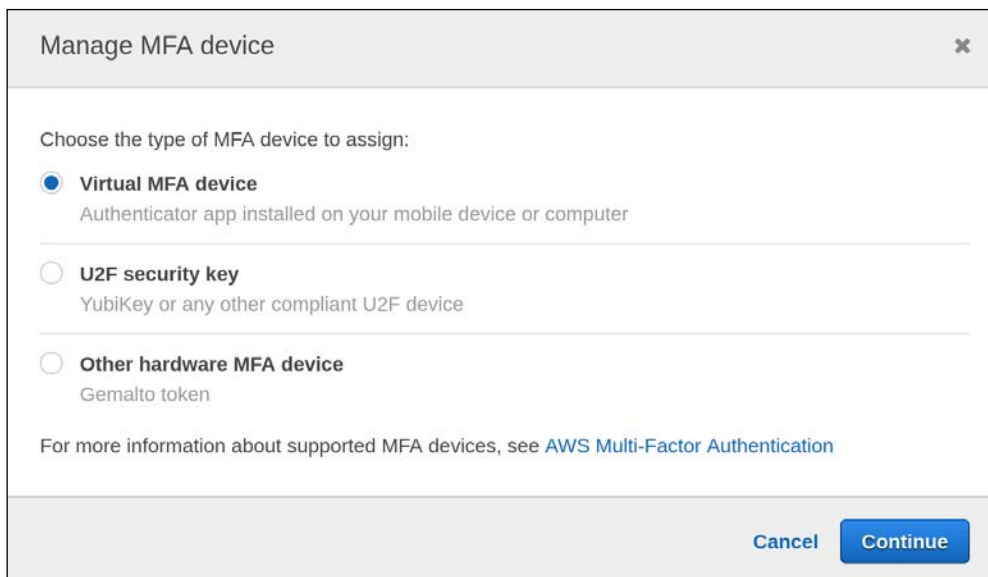



Figure 19.9: Choosing a type of MFA device to set up

To set up your virtual MFA device, you'll need to download a second-factor application to serve this purpose. Google Authenticator is a popular choice here, but I recommend Authy instead. Both are perfectly acceptable, but Authy also features a desktop app you can use, as well as the ability to recover your account if your primary device is not accessible for any reason. Authy is compatible with Google Authenticator, so you can typically use it with services that offer a Google Authenticator option. To continue, reveal the QR code, scan it with your phone app, and then type in two subsequent values generated by the app.

Finally, click **Assign MFA** to finalize the process:

Set up virtual MFA device

1. Install a compatible app on your mobile device or computer
See a [list of compatible applications](#)
2. Use your virtual MFA app and your device's camera to scan the QR code

3. Type two consecutive MFA codes below
 MFA code 1:
 MFA code 2:

Alternatively, you can type the secret key. [Show secret key](#)

Cancel Previous **Assign MFA**

Figure 19.10: Setting up the MFA device

Now that we have two-factor authentication enabled for our root account, we should stop using that account immediately. This is actually a best practice when it comes to AWS; it's recommended that you create individual accounts for the individuals that will work on your AWS account, giving them the permissions they need to perform the tasks they need to complete. We'll discuss the **principle of least privilege** in *Chapter 21, Securing Your Server*, but it doesn't hurt to start thinking about the concept now. For now, what we'll do is create an administrator account for ourselves that we can use in place of the root account. We can always use the new account to create additional users if we need to do so.

To create a new administrator account, we will again utilize the IAM console. Once there, you'll see a **Users** link on the left, and then you can click the blue **Add users** button to begin the process:

Users (0) Info

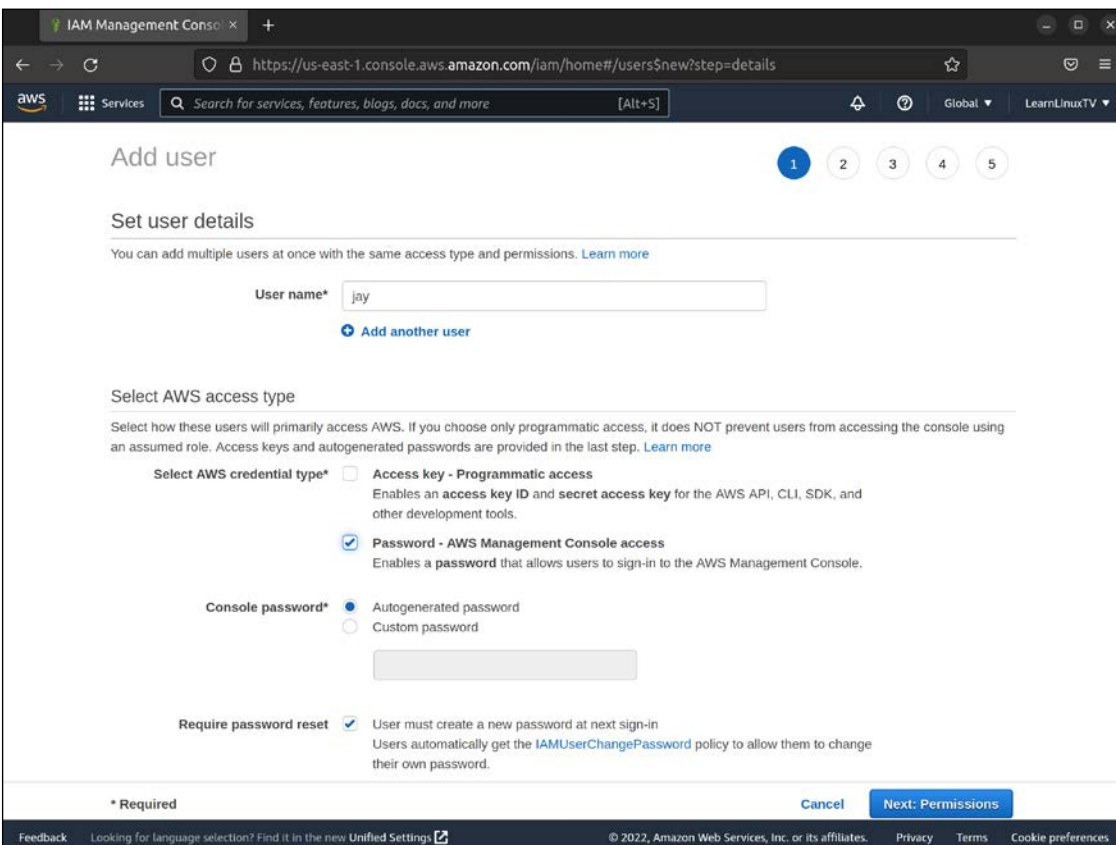
An IAM user is an identity with long-term credentials that is used to interact with AWS in an account.

Find users by username or access key

User name	Groups	Last activity	MFA	Password age	Active key age
No resources to display					

Figure 19.11: Setting up a new administrative user for AWS

The next screen will have you type the desired username, as well as setting **Access Type**. For the username, you can name the user whatever you'd like. For **Access Type**, we'll choose **AWS Management Console access**. Click **Next: Permissions** to continue:



The screenshot shows the AWS IAM console interface for adding a new user. The browser address bar indicates the URL is `https://us-east-1.console.aws.amazon.com/iam/home#/users$new?step=details`. The page title is "Add user" and it features a progress indicator with five steps, where the first step is active. The "Set user details" section includes a "User name*" field containing "jay" and a blue "Add another user" link. The "Select AWS access type" section provides instructions and two options: "Access key - Programmatic access" (unselected) and "Password - AWS Management Console access" (selected). The "Console password*" section offers "Autogenerated password" (selected) and "Custom password" (unselected). The "Require password reset" checkbox is checked, with a note that users will get the `IAMUserChangePassword` policy. At the bottom, there is a "Cancel" button and a blue "Next: Permissions" button. The footer contains "Feedback", "Looking for language selection? Find it in the new Unified Settings", "© 2022, Amazon Web Services, Inc. or its affiliates.", "Privacy", "Terms", and "Cookie preferences".

Figure 19.12: Setting up a new administrative user for AWS

Next, we will set up the appropriate permissions for our user. The third icon, labeled **Attach existing policies directly**, is the first selection we'll make here, and then below that we'll place a mark next to the check box for **Administrative Access**, and then we'll click **Next: Tags**:

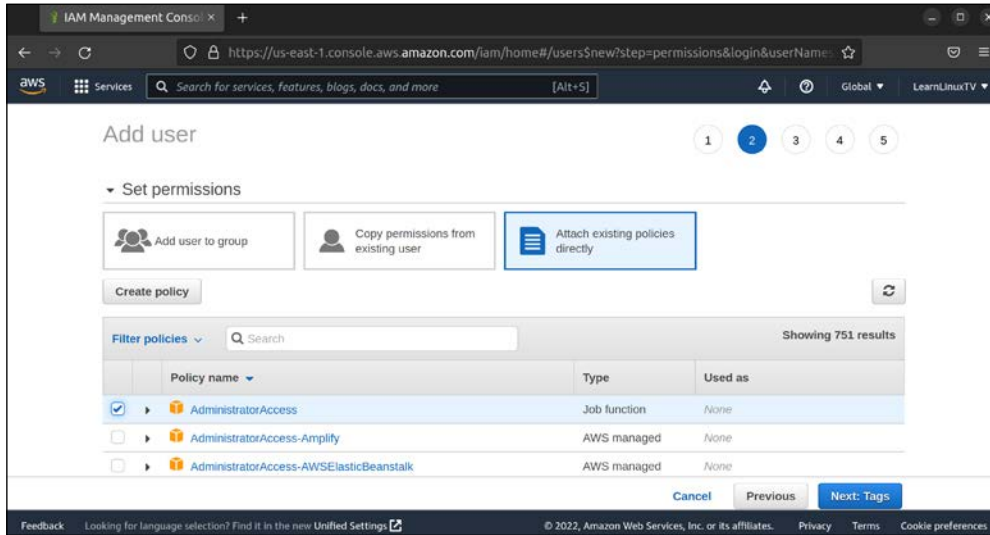


Figure 19.13: Setting up a new administrative user for AWS (continued)

We can skip the tags screen by leaving the fields blank and clicking **Next: Review**.

The next screen will provide us with an overview, and if everything appears to be correct here, we can click **Create user**:

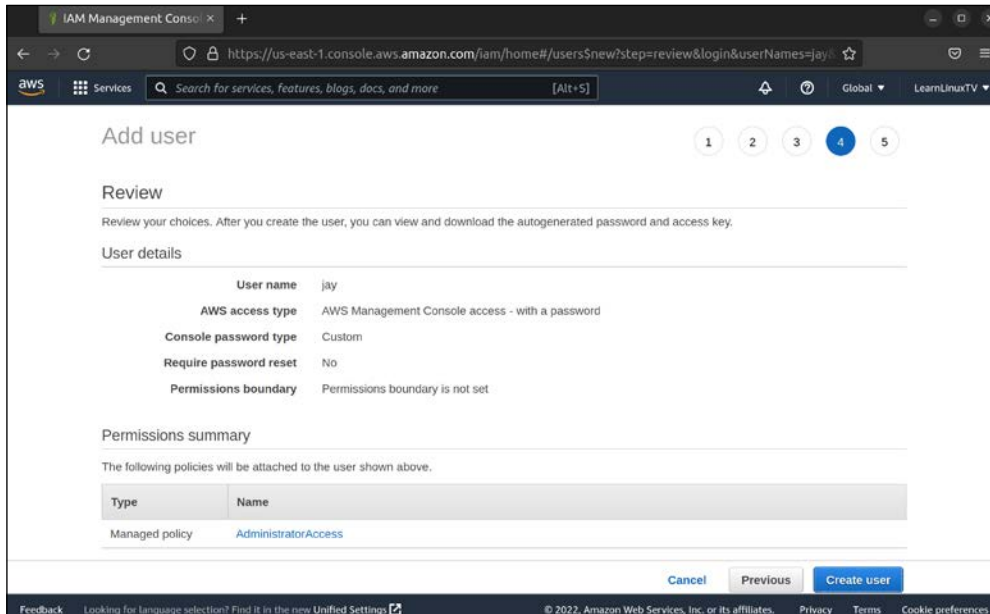


Figure 19.14: Setting up a new administrative user for AWS (continued)

Finally, we'll see a confirmation message that our user was created. If you chose to have a randomly generated password during the process, you should also see a button there to retrieve it:

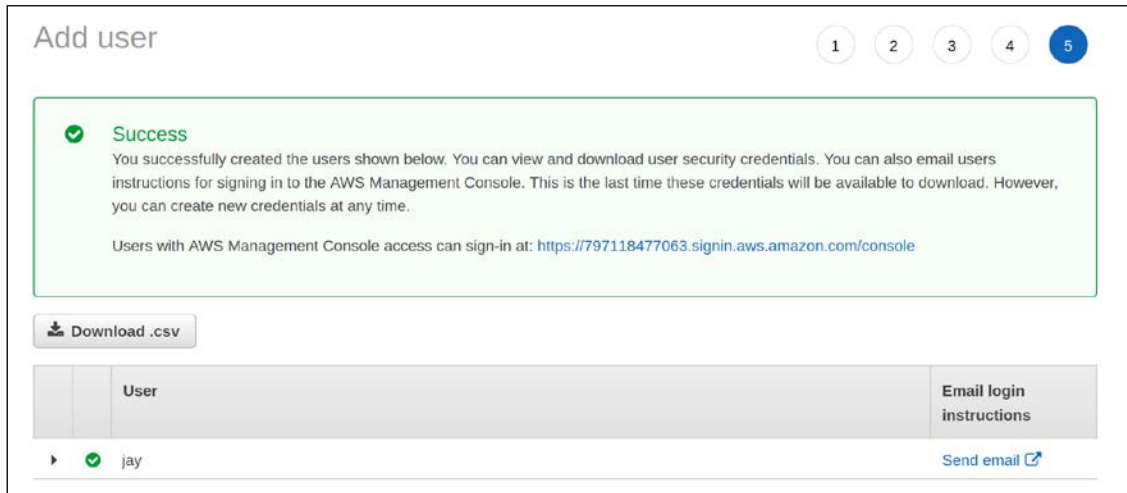


Figure 19.15: Setting up a new administrative user for AWS (continued)

You can now log in to the AWS Management Console with the new user and the password that was provided. You will also need the **account ID**, which is visible within the URL on this screen (it's the number right after `https://`). To make it easier, you can click on the URL that's shown in order to have the account ID autofilled.

Going forward, I recommend that you use the new user we've created for managing the AWS account, as it's a good practice to not use the root account unless you absolutely have to. In addition, I recommend you follow the procedure to set up MFA as we did earlier, but this time with the new user.

Choosing a region

As discussed earlier, a VPC within AWS is the high-level abstraction of your overall network. You can have multiple VPCs, which is similar to the concept of managing several physical networks. In fact, we already have VPCs created for us in our account, so we won't need to create one. In the future, keep in mind that creating additional VPCs is an option, should you ever need to have more than one. In our account, we have a default VPC in each **Region**, so choosing which one to utilize comes down to which region is most appropriate for our use.

For production use, you'll want to create instances in AWS that are as close to your customer as you can get. For example, let's say that the customers that your organization markets to are primarily located in the Eastern United States. There's a region available within AWS that is available that's labeled **US East**, so that would be an obvious choice in that scenario. You're not limited to regions within the USA though; there are regions available all over the world, such as in Germany, China, and Canada (among others). In a nutshell, you'll want to create resources as close to your customers as possible. If you don't have a preference, you can choose to utilize whichever region is closer to you.

Although it's beyond the scope of this book, AWS offers a service called **CloudFront** that acts as a **Content Delivery Network (CDN)** that you can use to make your resources available in various edge locations that users can be routed to in order to ensure they're retrieving your content from a location closest to where they are geographically. For organizations that produce media content, this is especially valuable. If this is something that might benefit you, I recommend reading more about CloudFront.

In addition, there are often multiple **Availability Zones** within various regions, which allow you to get even closer to your target audience. For example, when it comes to the US East region, there are two availability zones inside it, one in Virginia as well as another in Ohio. Availability zones not only give us the ability to get another step closer to our customers but also offer us additional options for redundancy. For example, if one availability zone goes down for whatever reason, you can route your customers to another. Availability zones have a specific naming syntax that consists of both the name of the region as well as the availability zone within that region. Using the Eastern United States as an example again, the two availability zones there are labeled **us-east-1** for Virginia and **us-east-2** for Ohio. Not all regions will have multiple availability zones, though. Canada currently only has one region with one availability zone: **ca-central-1**.

In addition to availability zones, there are also **local zones**, which are intended to allow you to set up resources even closer to your customers than availability zones are able to get. Local zones are a great choice if your application is sensitive to network latency, such as running a server for an online game. We won't go over local zones in this book at all, because this is a very new offering from AWS, and there are only two of them in existence as of the time this book is being prepared for publishing. Amazon intends to add additional local zones in the future, so there may be more of them available by the time you're reading this. If your organization offers a service that is sensitive to network latency, this may be a feature you'll want to keep up to date on as they roll it out to more locations.

For now, the only consideration is which region will benefit your customers by being as close to them as possible. When it comes to following along with the examples in this book though, choosing a region closest to you geographically is a good idea.

Now that we've selected our region, how about we create an actual Ubuntu instance in the cloud? That's exactly what we'll do in the next section.

Deploying Ubuntu as an AWS EC2 instance

With a great deal of discussion out of the way, it's time to create an actual Ubuntu deployment in the cloud. This will allow us to see the AWS service in action and give us some working experience with the EC2 service. This requires two individual steps: the first to create a required IAM role and the second to create our instance. Let's first make sure we understand the requirements of the IAM role, then we'll set up the role and then create our new instance.

Setting up an IAM role for Session Manager

Session Manager is a service within AWS that we can use to access a command prompt for our instance. It's actually part of **Systems Manager** and not its own service. If you want to access Session Manager, you will need to search for Systems Manager, and you'll find Session Manager as a service underneath that. You'll see this shortly.

Why should we use Session Manager? Just like with any other Linux server, we can still use OpenSSH to connect to the EC2 instance we'll be creating, just as we have many times while working with non-AWS instances throughout this book.

There's nothing wrong with using OpenSSH; with the right settings it can be a very secure option. In fact, we will explore methods of better securing it in *Chapter 21, Securing Your Server*. With AWS, we can use Session Manager as an alternative to OpenSSH, and it's a worthwhile alternative to learn that offers additional security in that its backend security is not something we have to manage ourselves. In addition, we can control access to it through the AWS console as well.

By default, Session Manager is not accessible at all. It requires a specific package to be installed within Ubuntu Server for it to work, and it also requires specific permissions to be enabled. The required package for Ubuntu is preinstalled by default, so the first requirement will be automatically taken care of for us immediately when we create our instance.

For the second requirement of adding permissions, we'll need to create an IAM role to allow the EC2 instance we're about to create to communicate with the Session Manager service. When I mentioned earlier that Session Manager isn't accessible by default, this is why—it's missing the permissions needed until we add them. To add the required permissions, we'll access the IAM service within the AWS console, the very same one that we used earlier to create a user account for ourselves. IAM itself has many tricks up its sleeves, more than just simply allowing us to create users. It also allows us to create **IAM roles**, which give us the ability to add permissions to entire objects. For example, we can create a role with the permissions that are required, and then we can attach that role to any EC2 instance to immediately give it the ability to be connected to by Session Manager.

Let's get started and set up the required IAM role for Session Manager. Return to the IAM section of the AWS console that we've worked with a few times now, and we'll create the required role.

In the IAM menu on the left side of the window, click on **Roles**, and then click on the blue button labeled **Create role**:

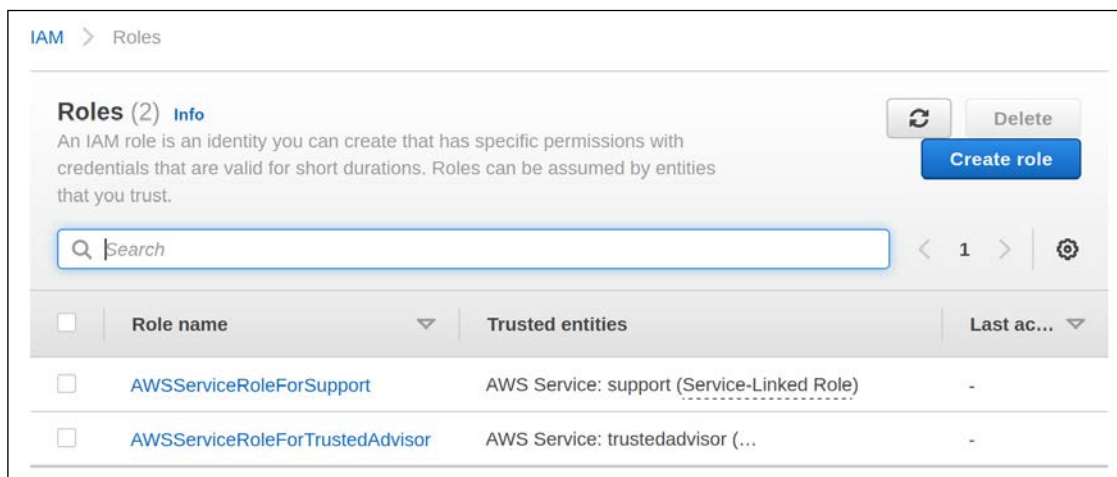


Figure 19.16: Creating an IAM role to enable Session Manager

On the next screen, make sure **AWS service** is selected, and in the menu below that, choose **EC2** as the service. Click **Next** to continue along:

Select trusted entity

Trusted entity type

AWS service
 Allow AWS services like EC2, Lambda, or others to perform actions in this account.

AWS account
 Allow entities in other AWS accounts belonging to you or a 3rd party to perform actions in this account.

Web identity
 Allows users federated by the specified external web identity provider to assume this role to perform actions in this account.

SAML 2.0 federation
 Allow users federated with SAML 2.0 from a corporate directory to perform actions in this account.

Custom trust policy
 Create a custom trust policy to enable others to perform actions in this account.

Use case

Allow an AWS service like EC2, Lambda, or others to perform actions in this account.

Common use cases

EC2
 Allows EC2 instances to call AWS services on your behalf.

Lambda
 Allows Lambda functions to call AWS services on your behalf.

Use cases for other AWS services:

Choose a service to view use case ▼

Cancel
Next

Figure 19.17: Creating an IAM role to enable Session Manager (continued)

Next, we are able to attach policies to our role. In the search field that appears, we can type a keyword to narrow down the list, and then click on a checkbox next to a policy we wish to attach to our role. A full overview of all of the built-in policies and what they're for is beyond our scope, but as a short summary, each service within AWS has pre-built policies that can be attached to a role, which allows access to various features. Specific to our needs, we will add the **AmazonSSMFullAccess** policy to the role we're creating.

The purpose of this will become clearer when we create our EC2 instance, so for now, click **Next** to continue on:

Add permissions

Permissions policies (Selected 1/751) Refresh Create Policy

Choose one or more policies to attach to your new role.

Filter policies by property or policy name and press enter 1 match < 1 > Settings

"amazonssmfull" × Clear filters

<input checked="" type="checkbox"/>	Policy name ↗	Type	Description
<input checked="" type="checkbox"/>	+ AmazonSSMFullAc...	AWS m...	Provides full access to Amazon SSM.

▶ Set permissions boundary - optional

Set a permissions boundary to control the maximum permissions this role can have. This is not a common setting, but you can use it to delegate permission management to others.

Cancel Previous Next

Figure 19.18: Attaching the AmazonSSMFullAccess policy to our role

The next screen that will appear will give us the ability to add one or more tags. We'll skip this for now, but you can feel free to add any tags you'd like here. Tags allow you to attach information to a resource and aren't limited to IAM roles. You can add any descriptive information you feel is pertinent, if you wish. Tags are simply **key: value** pairs, so there's no specific naming scheme to follow here. Add tags if you wish to do so, and when you're finished with this screen click **Next: Review**.

The final screen will give us a review of the settings we've chosen so far, as well as an option to name the role, and add a description if we wish to do so. Although it's optional, I recommend giving the role a name, to make it easier to identify later. When you're finished, click **Create role**:

Step 2: Add permissions Edit

Permissions policy summary

Policy name ↗	Type	Attached as
AmazonSSMFullAccess	AWS managed	Permissions policy

Tags

Add tags (Optional)
Tags are key-value pairs that you can add to AWS resources to help identify, organize, or search for resources.

No tags associated with the resource.

Add tag
You can add up to 50 more tags

Cancel Previous Create role

Figure 19.19: Adding a name and description to our role

When it comes to setting up our IAM role, we're all set—the role has been created and we can go ahead and use it. Next, it's time to create our Ubuntu instance.

Creating an Ubuntu Server instance in AWS

Now it's time to see our work come together and create our Ubuntu instance. In the AWS console, we should first access the EC2 service to get started. You can easily find any service by typing its name into the search box within the console; so if you start typing EC2 into that field, you should see EC2 on the list. After you click on that, click on **Instances** on the left side of the screen. After doing that, you'll see a screen with a button labeled **Launch instances**:

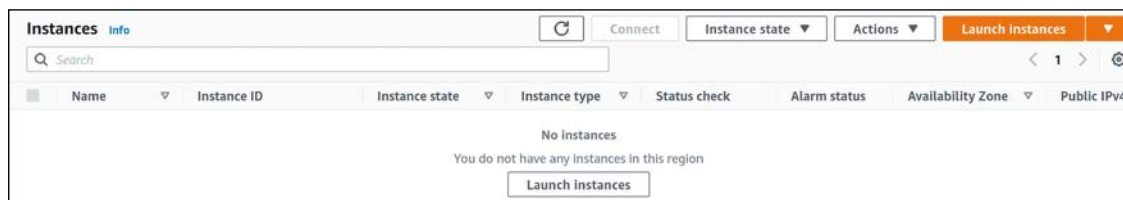


Figure 19.20: The main window of the EC2 service

Normally, the **Instances** section of the EC2 console will show us a list of all of our server instances, but unless you've read ahead, we don't have any yet so the window is blank. When you click on **Launch instances**, you'll see various operating systems in the list. But for our purposes, Ubuntu is shown in the **Quick Start** section, so we'll select that:

Name and tags [Info](#)

Name

ubuntu-server [Add additional tags](#)

▼ **Application and OS Images (Amazon Machine Image)** [Info](#)

An AMI is a template that contains the software configuration (operating system, application server, and applications) required to launch your instance. Search or Browse for AMIs if you don't see what you are looking for below

🔍 Search our full catalog including 1000s of application and OS images

Quick Start

Amazon Linux **Ubuntu** Windows Red Hat SUSE Linux

aws ubuntu® Microsoft Red Hat SUSE

[Browse more AMIs](#)

Including AMIs from AWS, Marketplace and the Community

Amazon Machine Image (AMI)

Ubuntu Server 22.04 LTS (HVM), SSD Volume Type Free tier eligible ▼

ami-09d56f8956ab235b3 (64-bit (x86)) / ami-02ddaf75821f25213 (64-bit (Arm))

Virtualization: hvm ENA enabled: true Root device type: ebs

Figure 19.21: The Ubuntu option for EC2

Further down, make sure that **t2.micro** is selected, which should show the verbiage **Free tier eligible** alongside it. That's the instance type that we'll want to use:

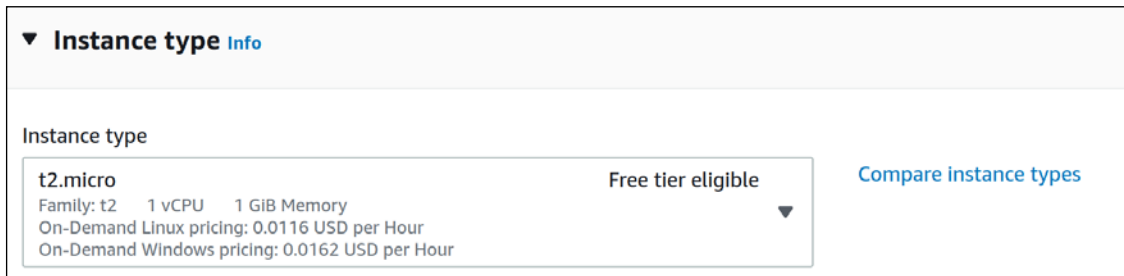


Figure 19.22: Choosing an instance type

On the **Choose an Instance Type** screen, there will be quite a few instance types from which to choose. I selected the **t2.micro** instance type, and I recommend you do the same. It's eligible for the free tier, which is a special tier you'll have access to within the first 12 months of the age of the account. You most likely wouldn't choose this instance type for a production server, as it will be quite slow—it only has 1 CPU and 1 GiB of memory. And it's a burstable instance type, which means the speed fluctuates based on usage. It's able to burst to take care of busy workloads, but its ability to do so depends on CPU credits that it earns in a particular time. A full explanation is beyond the scope of this chapter, but if you're going to use AWS in production, it's a good idea to read about the various instance types available. Although you don't see it in the screenshot, you should also see the cost on this page as well. But again, we'll utilize the free tier for now.

Further down the page, you'll generate a new key pair, assuming you don't have one already. This will be used for connecting to the instance via SSH. To generate a new key, click **Create new key pair**:



Figure 19.23: Key pair settings during instance creation

The next screen that appears will allow us to configure the key, such as setting encryption options. For our purposes, we can leave the defaults as is after giving the key a name, and then we can click **Create key pair**:

Create key pair ✕

Key pairs allow you to connect to your instance securely.

Enter the name of the key pair below. When prompted, store the private key in a secure and accessible location on your computer. **You will need it later to connect to your instance.** [Learn more](#)

Key pair name

The name can include upto 255 ASCII characters. It can't include leading or trailing spaces.

Key pair type

RSA
RSA encrypted private and public key pair

ED25519
ED25519 encrypted private and public key pair (Not supported for Windows instances)

Private key file format

.pem
For use with OpenSSH

.ppk
For use with PuTTY

Cancel Create key pair

Figure 19.24: Customizing our SSH key

The previous step will trigger your browser to download the SSH key that you've generated. Be sure to keep it in a safe place. You can use the key you've just downloaded to connect to your instance via OpenSSH with a command similar to the following:

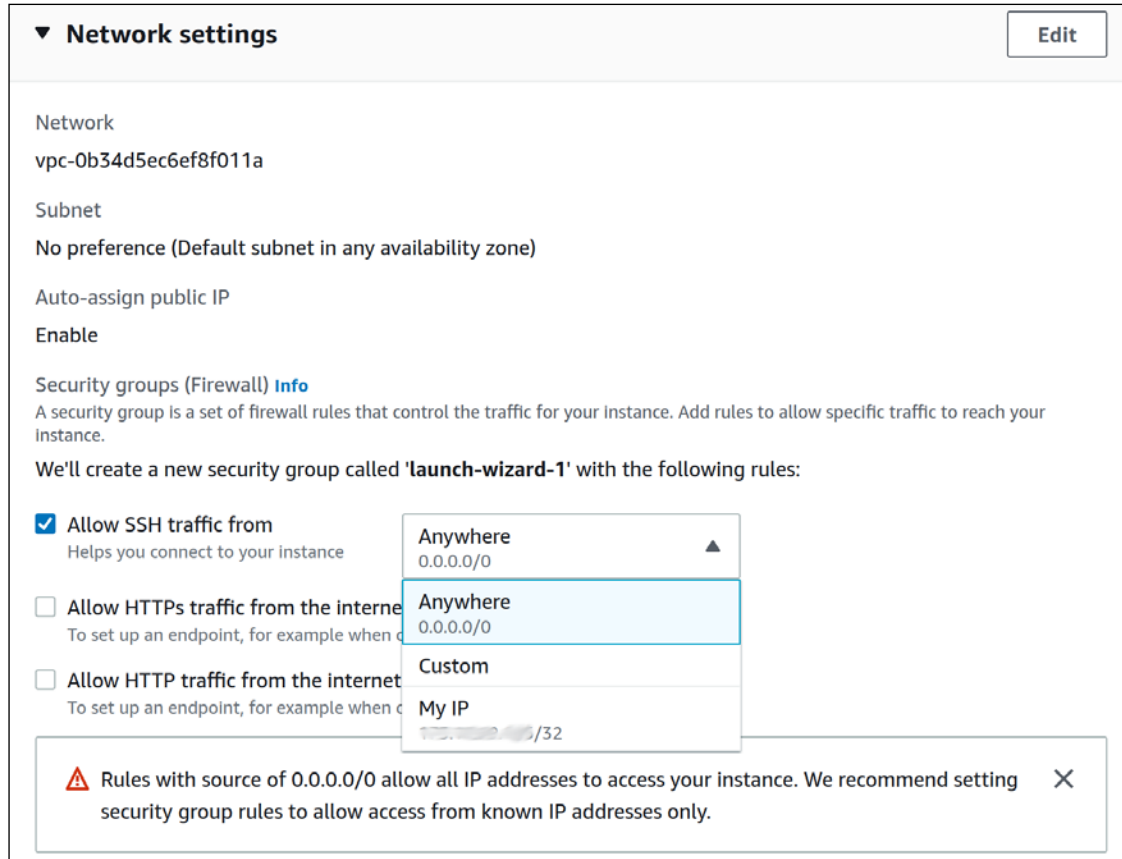
```
ssh -i /path/to/key.pem ubuntu@<Instance Public IP>
```

For me, if I add the path of my key as well as the public IP address listed for my instance, the command becomes this:

```
ssh -i /home/jay/downloads/jay_ssh.pem ubuntu@54.81.234.225
```

From here, managing the server is just a matter of interacting with its shell and entering commands.

But anyway, I'm getting ahead of myself. After you safeguard your key, we can move on to the next section by scrolling down. Network settings are what we'll work on next:



Network settings Edit

Network
vpc-0b34d5ec6ef8f011a

Subnet
No preference (Default subnet in any availability zone)

Auto-assign public IP
Enable

Security groups (Firewall) [Info](#)
A security group is a set of firewall rules that control the traffic for your instance. Add rules to allow specific traffic to reach your instance.

We'll create a new security group called **'launch-wizard-1'** with the following rules:

- Allow SSH traffic from**
Helps you connect to your instance
- Allow HTTPs traffic from the internet**
To set up an endpoint, for example when c
- Allow HTTP traffic from the internet**
To set up an endpoint, for example when c

Source options: Anywhere 0.0.0.0/0, Anywhere 0.0.0.0/0, Custom, My IP

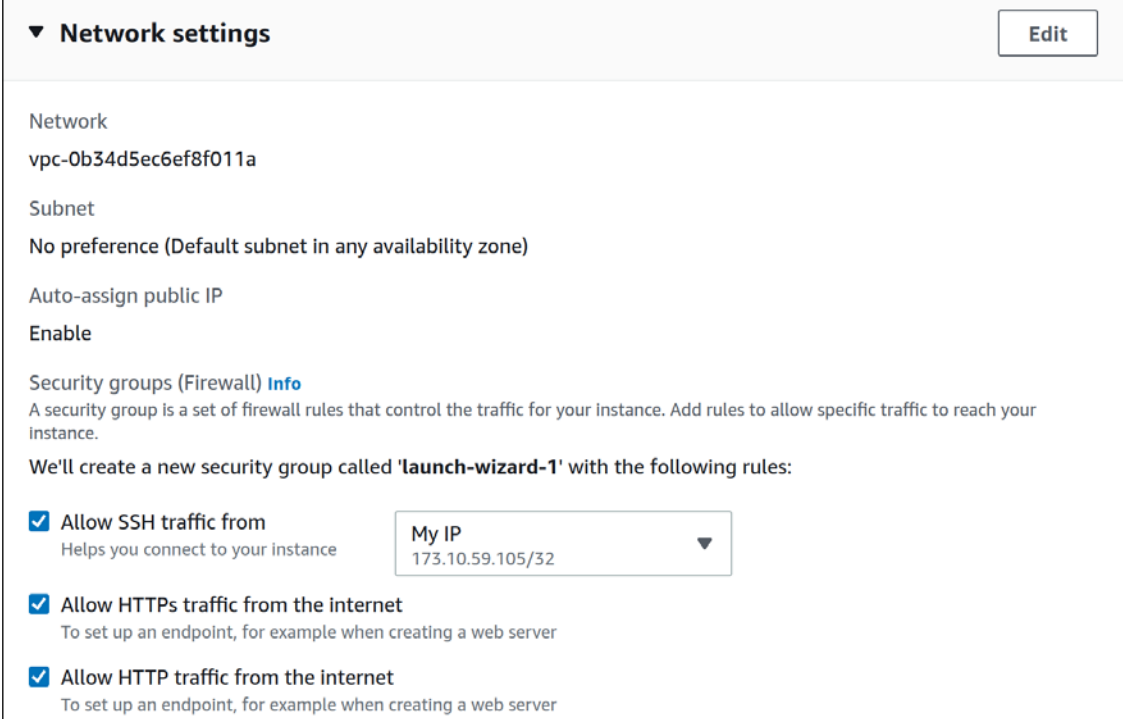
Warning: Rules with source of 0.0.0.0/0 allow all IP addresses to access your instance. We recommend setting security group rules to allow access from known IP addresses only.

Figure 19.25: Setting options for our new EC2 instance

The first consideration on this screen is where we'll allow OpenSSH connections from. The default option, `0.0.0.0/0`, allows connections from anywhere (and is even labeled as such). As we'll discuss in *Chapter 21, Securing Your Server*, allowing SSH connections publicly is a bad idea, since outside threat actors might try to use that as a means to access the server. With the instance we're creating right now, we can make a case for allowing connections from anywhere, since it's a test instance for the purposes of learning (and we'll be deleting the instance before the close of the chapter). However, it might be a better idea to get into the habit of restricting access to OpenSSH to specific IP addresses. To do that, you can choose the option **My IP** from the dropdown, which will restrict access to OpenSSH to your public IP address.

If you do choose to restrict access, keep in mind that unless your public IP is static, it can change at any time. When it does, you'll lose access to your EC2 instances via OpenSSH. You can regain access by updating the security group for the instance to include your new IP address, but that inconvenience is a small price to pay for added security benefit.

Continuing in that same section, we have an option to allow traffic in addition to OpenSSH:



The screenshot shows the 'Network settings' section of an AWS console. It includes an 'Edit' button in the top right. The settings are as follows:

- Network:** vpc-0b34d5ec6ef8f011a
- Subnet:** No preference (Default subnet in any availability zone)
- Auto-assign public IP:** Enable
- Security groups (Firewall):** Info. A security group is a set of firewall rules that control the traffic for your instance. Add rules to allow specific traffic to reach your instance. We'll create a new security group called 'launch-wizard-1' with the following rules:
 - Allow SSH traffic from** My IP (173.10.59.105/32). Helps you connect to your instance.
 - Allow HTTPS traffic from the internet**. To set up an endpoint, for example when creating a web server.
 - Allow HTTP traffic from the internet**. To set up an endpoint, for example when creating a web server.

Figure 19.26: Network settings while creating an EC2 instance on AWS

The other options, **HTTPS** and **HTTP** respectively, are essential if you're planning on setting up a website on the instance. It's very common to restrict OpenSSH on a web server, while at the same time allowing full access to ports 80 and 443 from the public internet. The reason for this is, OpenSSH gives you access to manage the server, so we definitely wouldn't want that open to the public. When it comes to a website, we'll most likely want the general public to be able to access it, which is why we allow those options. You should only check the latter two boxes in the situation where you're actually planning on setting up a web server. For our needs, we should allow these boxes, since we will be installing Apache later.

Next, we'll configure the **storage** for our instance:

▼ **Configure storage** [Info](#) Advanced

1x 30 GIB gp2 Root volume

Free tier eligible customers can get up to 30 GB of EBS General Purpose (SSD) or Magnetic storage

[Add new volume](#)

The selected AMI contains more instance store volumes than the instance allows. Only the first 0 instance store volumes from the AMI will be accessible from the instance

0 x File systems [Edit](#)

Figure 19.27: Storage options while creating an EC2 instance

By default, your EC2 instance will be set up with 8 GiB of storage space. As indicated in the highlighted message on the screen, you can get up to 30 GB of EBS storage space as part of the free tier, if you're eligible. For that reason, you can consider setting this higher. You can also add additional storage volumes to the instance, which might help if you wanted to segregate storage between block storage devices. We shouldn't need that for our use case in this chapter.

Below that, the final section on this screen allows you to configure **Advanced details**. There are quite a few extra options here, but we'll be ignoring all but one of them. At the very bottom, there's a large text box labeled **User data**. This is the only option within the advanced details section we'll change:

User data [Info](#)

```
#!/bin/bash
apt update
apt dist-upgrade -y
apt install -y apache2
```

Figure 19.28: Adding user data for our EC2 instance

The **User data** section is often overlooked, but it's *incredibly* useful. But what exactly is this for, and what does it do? If I didn't know any better myself, I'd probably assume that this is where you'd add information that pertains to your users. But that's not what it's for. What **User data** allows you to do is add a series of commands, or even a full script, that you'd like to have executed as the instance is created.

Already, you can think of any command you might run, or configuration you might tweak, any time you set up a server. Instead of manually executing setup commands after the instance is online, you can simplify this quite a bit and have much of that done automatically before you even log in for the first time.

In my case, I've added the following code to the **User data** field:

```
#!/bin/bash
apt update
apt dist-upgrade -y
apt install -y apache2
```

If you think the code that I've added resembles a Bash script, then you're correct—that's exactly what it is. I added four lines of Bash statements to the **User data** field, to have some commands run automatically. The code should be relatively straightforward: I have it set up to update the repository index, then perform a full system upgrade.

This is important; we always want our servers to start with the latest patches available. As a proof of concept, I added a statement to install Apache. Notice that I included the `-y` option to all of the `apt` commands. This automatically responds “yes” to any question `apt` may ask, since we don't have a display hooked up to this server and are unable to answer questions ourselves. Without that option, the user data will fail to apply.

Anyway, the moment of truth is here – it's time to launch our instance, and we can do so by clicking the **Launch instance** button near the bottom of the page. Assuming we didn't forget anything along the way, our new Ubuntu Server cloud instance is moments away from existing!

At this point, we should see the new instance in the list of EC2 instances in our account, and it will take some time for it to be ready for use. When the status indicates that the instance is **Running**, then that means it's ready for us to connect to:

Name	Instance ID	Instance state	Instance type	Status check	Alarm status	Availability Zone
ubuntu-server	i-0a6653f8295e720e8	Running	t2.micro	2/2 checks passed	No alarms	us-east-1b

Figure 19.29: Checking the status of our EC2 instance

In order to set up a connection so we can manage the instance, we'll need to change the **IAM role** for the instance. If you recall, we created an IAM role earlier, and this is where we'll attach that role to our new instance. This will ensure that the settings within the role are applied to the instance, which in turn will give us access to Session Manager for connecting to that instance. To change the IAM role, we'll check the box to the left of the instance, then click on **Actions** at the top of the screen. In the menu that opens, click on **Security**, and then **Modify IAM role**:

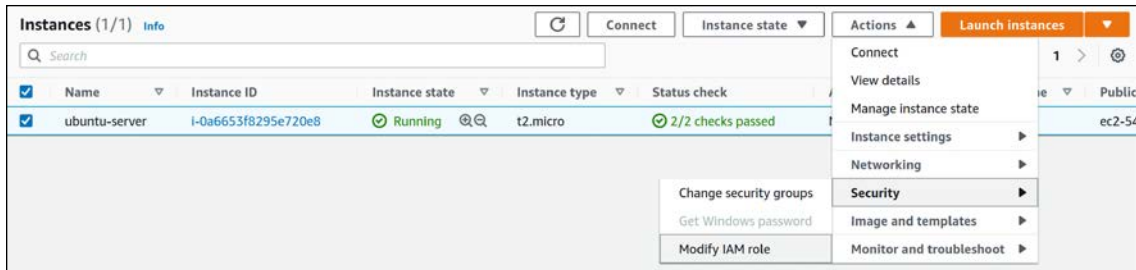


Figure 19.30: Navigating to the IAM role settings for an EC2 instance

The screen that follows is where we'll change the IAM role to the one we created earlier:

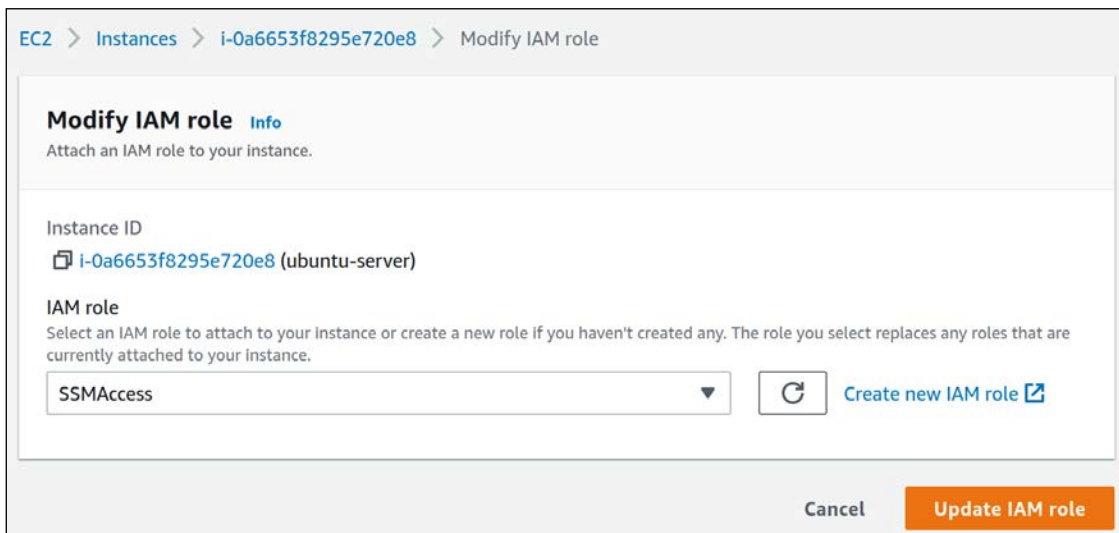


Figure 19.31: Changing the IAM role of an instance

After you click **Update IAM role**, you'll be brought back to the list of current EC2 instances. If you click on the blue instance ID text near the name of the instance, you'll be taken to a screen where you can customize additional options for the instance:

The screenshot displays the 'Instance summary' for an EC2 instance. The instance is named 'i-0a6653f8295e720e8 (ubuntu-server)' and is currently in a 'Running' state. The summary is organized into three columns:

- Column 1:** Instance ID (i-0a6653f8295e720e8), IPv6 address (none), Hostname type (IP name: ip-172-31-30-208.ec2.internal), and Instance type (t2.micro).
- Column 2:** Public IPv4 address (54.88.57.142), Instance state (Running), Private IP DNS name (ip-172-31-30-208.ec2.internal), and Elastic IP addresses (none).
- Column 3:** Private IPv4 addresses (172.31.30.208), Public IPv4 DNS (ec2-54-88-57-142.compute-1.amazonaws.com), Answer private resource DNS name (IPv4 (A)), and Auto-assigned IP address (54.88.57.142 [Public IP]).

Figure 19.32: Additional EC2 instance settings

Among the various items on the page, of special interest to us is the **Connect** button. If we click on that, we'll begin the process of setting up a remote connection we can use to manage the instance. On the screen that appears, click on the **Session Manager** tab:

The screenshot shows the 'Connect to instance' page. The page title is 'Connect to instance' and it provides instructions on how to connect to the instance. There are four tabs for different connection methods: 'EC2 Instance Connect', 'Session Manager' (which is selected and highlighted in orange), 'SSH client', and 'EC2 Serial Console'. Below the tabs, there is a section titled 'Session Manager usage:' with the following bullet points:

- Connect to your instance without SSH keys or a bastion host.
- Sessions are secured using an AWS Key Management Service key.
- You can log session commands and details in an Amazon S3 bucket or CloudWatch Logs log group.
- Configure sessions on the Session Manager [Preferences](#) page.

At the bottom right of the page, there are two buttons: 'Cancel' and 'Connect'.

Figure 19.33: Preparing a session to connect to an instance

Once you click the **Connect** button, a new browser window will appear that you can use to issue commands to your instance:

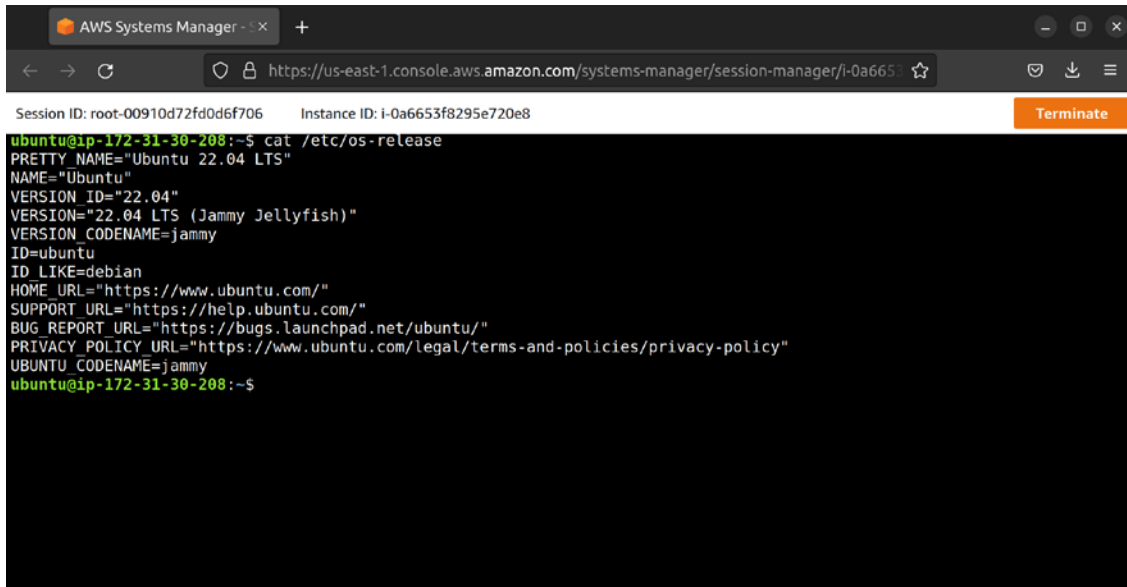


Figure 19.34: An instance session open within a web browser

In Figure 19.34, I entered the following command in order to display details for the version of Ubuntu that was deployed in the instance:

```
cat /etc/os-release
```

The `/etc/os-release` file is included with all Ubuntu installations, and as you can see from the output, it contains some information regarding the version of Ubuntu we're running. That command was entered directly into the Session Manager window, to show that the connection is actually working and we can now configure the instance right from within our web browser!

If you recall, we added **User Data** earlier, and that included a command to install Apache. If you enter the public IP address of your EC2 instance into a web browser, you should see the default Apache web page:

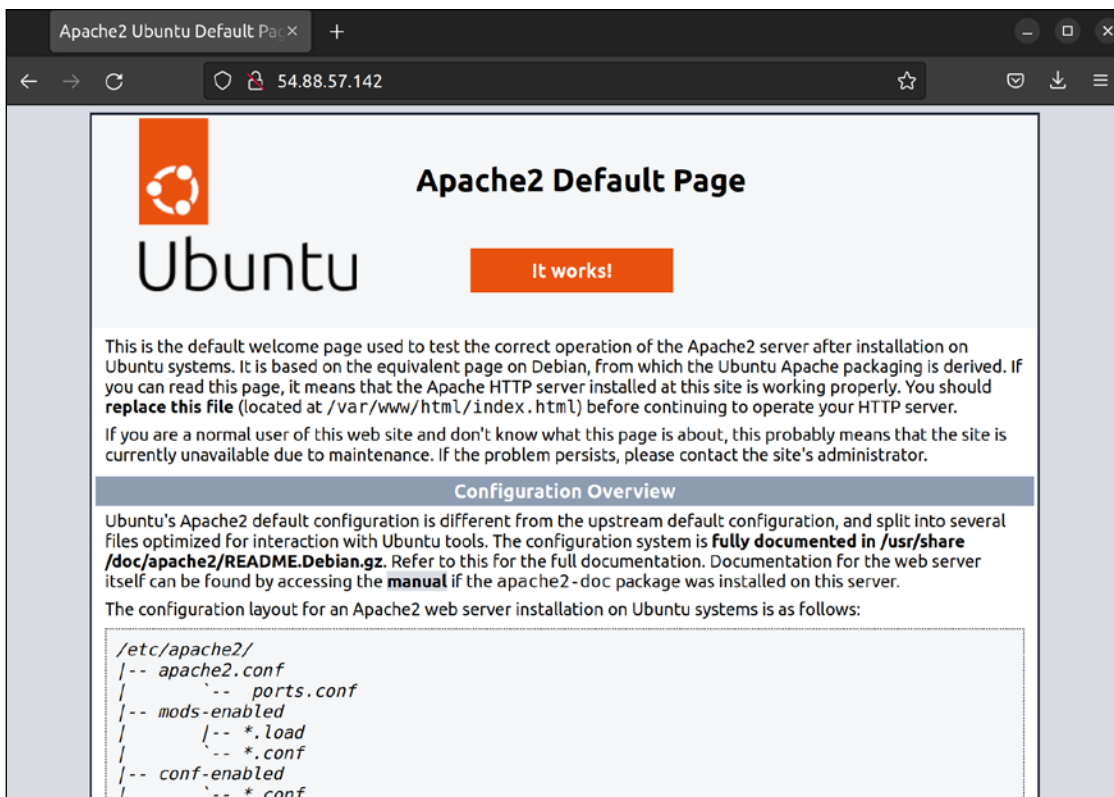


Figure 19.35: The Apache default web page running on an EC2 instance

Congratulations! You've successfully deployed Ubuntu Server to the cloud, and now have an actual web server running on it. That's all there is to it. Using Session Manager is also simple; all you need to do to customize the server further is right-click on it, click **Connect**, and you can then continue to build the instance. That's awesome!

What's not so awesome, though, is when something happens to your server and you have to start over and rebuild it from scratch. In the next section, we're going to explore the process of creating an image of the server that we can utilize to deploy customized versions of Ubuntu.

Creating and deploying Ubuntu AMIs

Just about every cloud platform I know of includes some sort of feature that can be used to create images of the instance's hard disk. An image can be used to create copies of the original server, as well as acting as a starting point so if the server needs to be rebuilt, we won't have to start over from scratch. In AWS, images are known as **Amazon Machine Images (AMIs)**. For all intents and purposes, there's nothing very unique about AMIs; if you've worked with disk images in the past, it's the same thing. When it comes to what to include in an AMI, you can (and should) use your imagination here—anything you find yourself manually setting up or configuring while rolling out a new server is a candidate to be included in an image, and the more customizations you include inside the image, the more time it will save you later.

Let's see this in action and create an image of the server we've just set up. We should consider shutting down our server first, although this isn't required. Taking an AMI of a server that is shut down is preferred over doing the same on a server that is running. When the server is shut down, nothing is writing to its disk, so we don't have to worry about corruption if we're capturing an AMI in the middle of a critical write operation. The likelihood of running into an issue while creating an AMI of a running server is very small, but I recommend shutting down the server if you can just to be on the safe side.

In the EC2 console of AWS, you can right-click on the instance to access **Session Manager**, and then you can simply power it off from the command prompt:

```
sudo poweroff
```

In AWS, it can take a minute or two for an instance to power down. You can refresh the page after some time, and the status should change to **Stopped**:

<input type="checkbox"/>	Name	Instance ID	Instance state	Instance type	Status check	Alarm status	Availability Zone
<input type="checkbox"/>	ubuntu-server	i-0a6653f8295e720e8	Stopped	t2.micro	-	No alarms	us-east-1b

Figure 19.36: Checking the status of an EC2 instance

Once the instance has stopped, you can right-click on it to begin the process of creating an AMI. Hover over **Image and templates** and then click **Create image**:

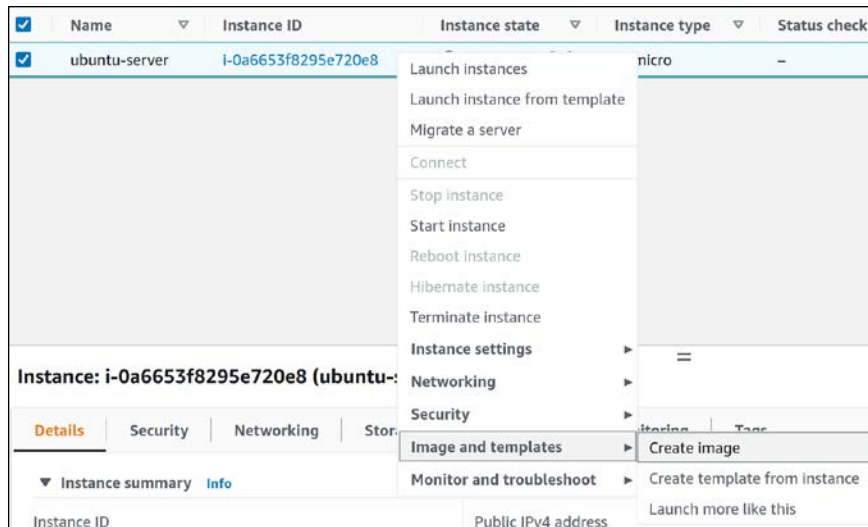


Figure 19.37: The Apache default web page running on an EC2 instance

Next, we can enter some details about our AMI. Give it a name and a description. This information will help others you work with understand what the image is for, and it can also help you remember why you've created the image later on down the road. There are other options on this page, but we can leave the rest as is. A name and description should be good enough for now. Once you're finished, click **Create image** to continue:

 A screenshot of the AWS console 'Create image' page. The 'Image name' field contains 'my-ubuntu-image' with a note: 'Maximum 127 characters. Can't be modified after creation.' The 'Image description - optional' field contains 'Initial setup of Apache' with a note: 'Maximum 255 characters.' Under 'No reboot', the 'Enable' checkbox is unchecked. The 'Instance volumes' section shows a table with columns: Volume type, Device, Snapshot, Size, Volume type, IOPS, Throughput, Delete on termination, and Encrypted. The first row has: EBS, /dev/sd..., Create new snapshot fr..., 30, EBS General Purpose SS..., 100, and Enable. There is an 'Add volume' button below. A blue box contains the text: 'During the image creation process, Amazon EC2 creates a snapshot of each of the above volumes.' The 'Tags - optional' section has a note: 'A tag is a label that you assign to an AWS resource. Each tag consists of a key and an optional value. You can use tags to search and filter your resources or track your AWS costs.' There are two radio button options: 'Tag image and snapshots together' (selected) and 'Tag image and snapshots separately'. Below this is the text 'No tags associated with the resource.' and an 'Add tag' button. At the bottom right, there are 'Cancel' and 'Create image' buttons.

Figure 19.38: Creating a new AMI

Believe it or not, that's all there is to it. The process of creating an AMI is very straightforward, with just a few steps. You should now see a confirmation screen, letting you know the image is in the process of being created:

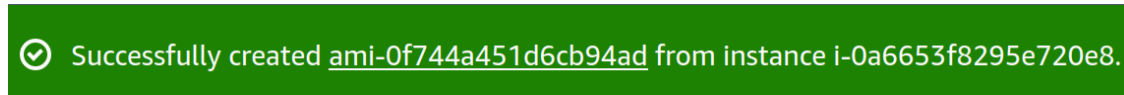


Figure 19.39: Confirmation while creating a new AMI

If you click on the underlined text that contains the AMI ID, you'll be directed to the AMI section of the EC2 console, where it will show your image on the list:

 A screenshot of the Amazon EC2 console's AMI section. The page title is "Amazon Machine Images (AMIs) (1) Info". There are buttons for "Recycle Bin", "EC2 Image Builder", "Actions", and "Launch inst...". A search bar is present with "Owned by me" and "Search" options. A filter bar shows "AMI ID = ami-0f744a451d6cb94ad" and a "Clear filters" button. Below is a table with columns: Name, AMI ID, AMI name, Source, and Owner. One row is visible with the following data:

Name	AMI ID	AMI name	Source	Owner
-	ami-0f744a451d6cb94ad	my-ubuntu-image	797118477063/my-ubuntu-image	797118477063

Figure 19.40: Our newly created AMI, available for use

In the AMI section of the EC2 console, you should see the list narrowed down to just the AMI we created just now. We only have one AMI anyway, unless you created multiple AMIs for practice. At the end, the **Status** column should read **available** if the AMI is ready for use. If not, give it some time, and refresh the page later. Sometimes it can take a few minutes. But with regard to creating an AMI, that's it!

Now that we have an AMI, how do we go about using it? Well, that's even easier actually. Simply right-click on the AMI on the list and click **Launch**. You'll see the same launch settings we worked through earlier when we originally created the instance, but this time, we're using our own AMI instead of the one provided to us. And now, we have our own custom AMI of Ubuntu with Apache built in that we can use to simplify our process a bit. Keep in mind, though, that our original instance is still stopped. You can return to your list of EC2 instances and start it by right-clicking on it, then clicking **Start**, but you don't have to; we're going to work through a fun automation example shortly.

In the next section, we're going to take a look at the concept of Auto Scaling.

Automatically scaling Ubuntu EC2 deployments with Auto Scaling

If we maintain one or more servers for our organization, it's hard to predict sometimes what the demand will be on that server. In the case of a popular news site, some articles may be more popular than others, and if something goes viral online, then requests to our site can increase by orders of magnitude in a short period of time. In the past, keeping up with customer demand was a very tedious process, one that may result in having to purchase an entirely new server with more powerful hardware. With our instance being in the cloud, we have more flexibility and can automate the process of bringing more servers online. And that's exactly what we're going to work on in this section.

Before we get started, keep in mind that we don't actually have a popular server in AWS; we only have a simple test server that's currently running Apache. We can simulate things to a point, but **Auto Scaling** is one of those things that requires a bit of practice to fully utilize. We will definitely get a working example here, though.

But another important thing to keep in mind is that the more instances we run, the higher the potential cost. We'll explore how to keep costs down in the next section, but as a general rule of thumb, delete whatever you're not using. As you've gone through examples in this chapter, we've set up our own EC2 instance. This is great: we were able to practice some concepts around AWS and put that to use. But if we leave something running that we don't need, we can have a surprise bill. It's a good idea to write yourself a reminder to delete everything in your test AWS account when we're done with the chapter, so you won't have to worry about that.

Continuing, one of the requirements of Auto Scaling is that we have an AMI ready that it will use to bring additional servers online. Since we've worked through creating an AMI in the previous section, we already have that requirement met. If you haven't already worked through the previous section, make sure you do so before we continue. The process of setting up Auto Scaling involves a handful of steps, and we'll work through each in their own subsection within this chapter.

Creating a launch template

Earlier in the chapter, we walked through the process of creating a new EC2 instance. We chose the option to launch an instance, and then configured various settings within multiple screens we worked through. We chose Ubuntu as our platform, added user data, and set an IAM role (among other things). What a **launch template** does is allow us to automate these choices. A launch template gives us the ability to automate the entire launch process.

On the left-hand menu of the EC2 console, there will be a link titled **Launch Templates**. Click on it. Once you do, you can click on the orange button labeled **Create launch template**. You'll then see a form you'll need to fill out, where you select all the defaults for the launch template. There's no screenshot on this page, because it's quite long and won't fit on one page. Instead, I'll include the relevant options below, with a short description and a recommendation regarding what to set the option to:

- **Launch template name:** This is simply a name for your launch template; set it to the name you feel is most appropriate. Note that this cannot contain spaces.
- **Template version description:** For the description, you can add some details that you think are relevant to the purpose of the launch template.
- **Amazon Machine Image (AMI):** Choose the AMI that you created in the previous section. You should see it listed if you click on the **My AMIs** tab, and ensure **Owned by me** is selected. This will narrow down the list of AMIs quite a bit, and since we've only created one AMI, it should be easy to find.
- **Key pair (login):** When you created the EC2 instance earlier, it had you create an OpenSSH key pair. If you drop down this list, that same key pair should be available. Choose that same key.
- **Instance type:** If you recall, we chose t2.micro as the instance type earlier. That's a good selection for this field as well, since t2.micro is eligible for the free tier.

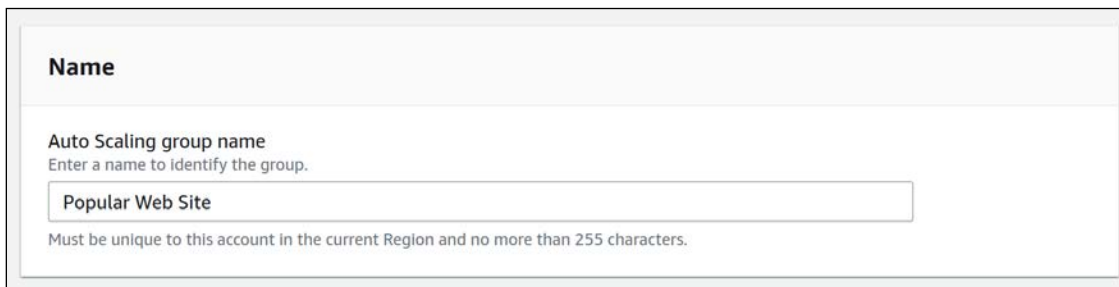
- **Security groups:** Earlier, when we added a security group, we set it up to allow OpenSSH and Apache. Feel free to choose that same security group for this.
- **IAM instance profile:** This option is a bit hidden, but you should see it as soon as you expand **Advanced details** near the bottom. Although this is optional, it's a good idea to select the IAM profile we created earlier, to ensure we have access to Session Manager for each instance the launch template creates.

With all of those details set, click **Create Launch Template** near the bottom of this screen. Now we have our launch template created and configured, and we can use it as part of the Auto Scaling feature we're in the process of building.

Creating an Auto Scaling group

Our next requirement is to create an **Auto Scaling group**, which will be a shorter process than setting up the launch template. An Auto Scaling group is a logical group of instances that are related to the overall application. We will add our launch template to this group and use it to customize requirements such as how many instances to have online.

Back in the EC2 dashboard, you'll find an option for creating Auto Scaling groups in the menu on the left side of the screen, closer to the bottom. Once there, give it a name:



Name

Auto Scaling group name
Enter a name to identify the group.

Popular Web Site

Must be unique to this account in the current Region and no more than 255 characters.

Figure 19.41: Naming the Auto Scaling group

Further down on that same screen, we will choose the launch template we created earlier. Go ahead and do so, and then click **Next**:

Launch template [Info](#) [Switch to launch configuration](#)

Launch template
Choose a launch template that contains the instance-level settings, such as the Amazon Machine Image (AMI), instance type, key pair, and security groups.

apache_launch_template_v1

[Create a launch template](#)

Version
Default (1)

[Create a launch template version](#)

Description	Launch template	Instance type
Initial version of the launch template	apache_launch_template_v1 lt-070d5478de348f4c7	-
AMI ID	Security groups	Request Spot Instances
ami-0f744a451d6cb94ad	-	No
Key pair name	Security group IDs	
production-key	sg-0963a41130e7193c7	

Additional details

Storage (volumes)	Date created
-	Thu May 26 2022 11:51:01 GMT-0400 (Eastern Daylight Time)

Figure 19.42: Selecting a launch template for our new Auto Scaling group

On the next screen, several additional sections worth of options will appear. The first of which will have you configure some defaults around networking. For the VPC, you can leave that as is, and for the availability zone, you can simply choose the first on the list:

Network [Info](#)

For most applications, you can use multiple Availability Zones and let EC2 Auto Scaling balance your instances across the zones. The default VPC and default subnets are suitable for getting started quickly.

VPC
Choose the VPC that defines the virtual network for your Auto Scaling group.

vpc-0b34d5ec6ef8f011a
172.31.0.0/16 Default

[Create a VPC](#)

Availability Zones and subnets
Define which Availability Zones and subnets your Auto Scaling group can use in the chosen VPC.

Select Availability Zones and subnets

us-east-1a | subnet-06bbb87a3ee976a78
172.31.80.0/20 Default

[Create a subnet](#)

Figure 19.43: Creating an Auto Scaling group (continued)

Continuing, the next section will ask us about our preference around instance types. On my end, I chose the **Manually add instance types** option, and then **t2.nano** as the instance type:

Instance type requirements [Info](#)

[Reset to launch template](#)

You can keep the same instance attributes or instance type from your launch template, or you can choose to override the launch template by specifying different instance attributes or manually adding instance types.

Specify instance attributes
 Provide your compute requirements. We fulfill your desired capacity with matching instance types based on your allocation strategy selection.

Manually add instance types
 Add one or more instance types. Any of the instance types may be launched to fulfill your desired capacity based on your allocation strategy selection.

Choose the instance types that best suit the needs of your application.

Primary instance type

1. 1vCPU 1 Gib Memory

Weight [Info](#)

ⓘ Your launch template does not specify an instance type. As a result, "Reset to launch template" cannot be chosen. You can continue by adding an instance type above.

Additional instance types

[Redo recommendations](#)

Add instance type

Figure 19.44: Creating an Auto Scaling group (continued)

Once you're finished choosing your instance type(s), there's nothing left for us to do on this particular page, so click **Next** to continue. This will take you to another section, this time asking you if you want to create a load balancer. In the first section of this screen, choose to **Attach to a new load balancer**:

Load balancing - optional [Info](#)

Use the options below to attach your Auto Scaling group to an existing load balancer, or to a new load balancer that you define.

No load balancer
 Traffic to your Auto Scaling group will not be fronted by a load balancer.

Attach to an existing load balancer
 Choose from your existing load balancers.

Attach to a new load balancer
 Quickly create a basic load balancer to attach to your Auto Scaling group.

Figure 19.45: Creating an Auto Scaling group (continued)

A **load balancer** allows us to route clients between multiple servers, so the end-users only see one endpoint, but behind the load balancer we can have more than one server available to serve client connections. This is going to be key to setting up our auto-scaled test site, so we'll definitely want to configure this.

After choosing to create a new load balancer, we can configure additional options in the next section. Most of these options we'll leave at their defaults, but there are a few things we'll want to pay attention to:

- **Load balancer type:** By default, **Application Load Balancer** should be selected. If it isn't, then make sure that's the highlighted option before continuing.
- **Load balancer name:** This is simply a name for your load balancer; set it to the name you feel is most appropriate. Note that this field is somewhat limited on what types of characters it will accept, so it's best to keep the name simple.
- **Availability Zones and subnets:** We'll need at least two availability zones for the load balancer, and one will be chosen for you already. The quickest way through this section is to enable the next availability zone in the list.

There are other options on this screen, but the next thing we'll be doing is creating a **target group**, which is directly underneath the availability zone options. There, we'll choose to create a **target group**:

The screenshot shows the 'Listeners and routing' configuration page in the AWS console. At the top, there's a heading 'Listeners and routing' and a note: 'If you require secure listeners, or multiple listeners, you can configure them from the [Load Balancing console](#) after your load balancer is created.' Below this, there are three main sections: 'Protocol' with a dropdown set to 'HTTP', 'Port' with a dropdown set to '80', and 'Default routing (forward to)' with a dropdown menu open. The dropdown menu has a search bar with a magnifying glass icon and a list item 'Create a target group' which is highlighted. Below these sections, there is a 'Tags - optional' section with a note 'Consider adding tags to your load balancer. Tags enable you...' and an 'Add tag' button.

Figure 19.46: Creating an Auto Scaling group (continued)

A **target group** is a logical grouping of EC2 instances that serve a common goal. A load balancer will send traffic to a target group, and instances within that group will answer the request. If an instance is not a member of a target group, then it won't be a part of our application. It's not uncommon for an organization to have more than one app, and as a result, several target groups. After you choose to create a target group, you can accept the default name if you wish, and then we should be finished with configuring our auto-scaling group. Click **Next** to continue, which will bring us to a section where we can configure our **scaling policy**:

Configure group size and scaling policies [Info](#)

Set the desired, minimum, and maximum capacity of your Auto Scaling group. You can optionally add a scaling policy to dynamically scale the number of instances in the group.

Group size - optional [Info](#)

Specify the size of the Auto Scaling group by changing the desired capacity. You can also specify minimum and maximum capacity limits. Your desired capacity must be within the limit range.

Desired capacity type
Choose the unit of measurement for the desired capacity value. By default, the desired capacity is measured in number of instances (units).

Units ▼

Desired capacity
1 ⬇ ⬆ ⬇

Minimum capacity
1 ⬇ ⬆ ⬇

Maximum capacity
1 ⬇ ⬆ ⬇

Figure 19.47: Creating an Auto Scaling group (continued)

This is where the magic happens: we can choose the minimum number of instances to have running at any one time and the maximum number of instances we will allow the application to scale up to. We can leave each field at 1 for now.

That should be about all we'll need to configure for our auto scaling group. You can click **Skip to review** to take a shortcut that will take you to the final screen of the process, which will show you an overview of your selections. If you're satisfied with the settings that are shown, you can finalize the process by clicking **Create Auto Scaling group**.

At this point, our Auto Scaling group is created, and we should have everything ready to go. At first, we'll have 0 instances within this group, so we'll see the current number of instances shown as 0.

After it finishes updating capacity, it will automatically spin up a new EC2 instance in order to meet our requirement of always having one instance online. If we check our list of EC2 instances, we should have a new one on the list now:

<input type="checkbox"/>	Name ▼	Instance ID	Instance state ▼	Instance type ▼	Status check
<input type="checkbox"/>	ubuntu-server	i-0a6653f8295e720e8	⊖ Stopped 🔍	t2.micro	-
<input type="checkbox"/>	-	i-07c806739a7ef33a7	⊕ Running 🔍	t2.nano	⊕ 2/2 checks passed

Figure 19.48: A new instance was created as part of our Auto Scaling configuration

As you can see in the screenshot, the original EC2 instance has a state of **Stopped**. We stopped it earlier so that we could create an AMI of that instance. In my case, I never started that instance again after creating the AMI, so it's completely stopped. The Auto Scaling configuration went ahead and created a new instance, because its requirement of having at least one instance running was not met.

Before we implement the final component we need for everything to function properly, let's take a moment to understand the value that we already have in place. If we were to increase the number of desired instances within the Auto Scaling group, then it would immediately spin up a new instance for us. Although advanced usage is more appropriate for a book that's dedicated to AWS, we can set this up to automatically happen when the CPU of our instance gets to a certain point, which can trigger Auto Scaling to bring another one online.

With just a single instance, we don't have Auto Scaling in play yet, but we can easily enable that. Another benefit that we get automatically is auto healing, and we have that benefit even with a single instance. If something were to happen to our only running instance, Auto Scaling will bring a new one online to replace it automatically.

In this situation, the website will be down for several minutes while the new instance is being created, but a bit of downtime is certainly better than having to manually replace the server ourselves. If we set the desired instances to a number higher than 1, a user will not notice anything if one of the servers goes down, the other will take care of the load while the new one comes up. These are amazing benefits to be able to take advantage of and will give us additional peace of mind right away. To test this yourself, feel free to delete the running EC2 instance by right-clicking it and then clicking **Terminate**. The instance should then get terminated, and a new one should appear within several minutes to replace it.

Since we have a load balancer in use, this changes the way that we'll access the application that's running on our instance(s), which in this case is Apache (we had Apache installed when we built our AMI, so every instance will automatically have Apache set up). Normally, we can access the application by navigating to the public IP address of the instance. We can still do this, actually – and the same is true for other instances within our load balancer setup as well. But in order to benefit from the overall solution we've built, we should instead access the application from the **DNS name** of the load balancer. This will ensure that our request is routed through the load balancer, rather than directly to a specific server. If you navigate to the **Load balancers** section of the management console, you should see the DNS name there. Going forward, the application or services you run on your instances should be served by having users access it via that DNS name.

Congratulations! At this point, you've created a complete load-balanced solution in AWS that will automatically heal from failures. Feel free to experiment a bit more, and then when you're finished, consider removing the test components we've created in this chapter to avoid a cost later on.

Speaking of cost, in the next section, we'll talk a bit further about how to manage costs and keep our bills under control.

Keeping costs down: understanding how to save money and make cost-effective decisions

As you just saw, there were many components and configurations we had to implement in order to build a load-balanced solution in AWS. As we grow our AWS infrastructure and implement more solutions, we should also keep an eye on our bill. Although we can utilize the free tier for now, production applications will likely need more powerful instances than what the free tier will provide, and the free tier itself won't last forever. Not only that, but we should also know how to check how our bill is trending to make sure we don't accidentally implement something that is expensive or waste money by running something we no longer need.

In this section, we'll explore some concepts around billing. Although it's beyond the scope of this chapter to do a complete deep dive into the world of billing, the subsections that follow will provide you with essential advice to help prevent unexpected charges.

Viewing billing information

Within AWS, the billing section is its own service along with others, such as EC2 and S3. You can find the **Billing** area from the service list. Once there, you'll be shown the **Billing & Cost Management Dashboard**, which will show you your current expenses and allow you to see current and past billing information:

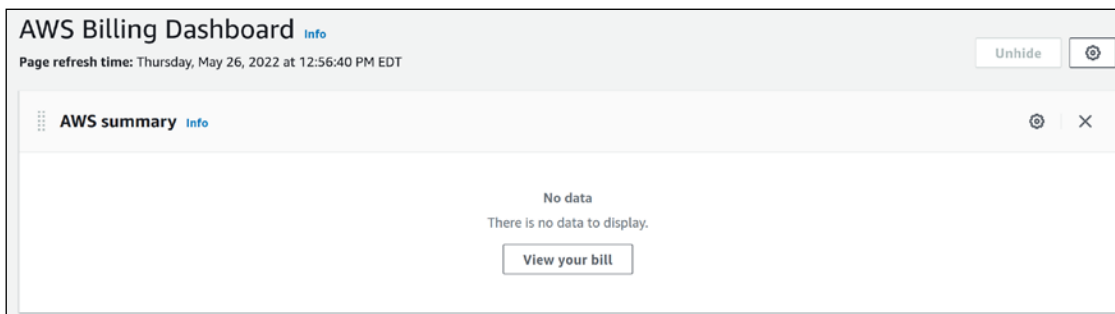


Figure 19.49: Viewing the Billing & Cost Management Dashboard

Since the account I'm working with currently was just created a week or so ago, and I'm only using instances in the free tier, I have no costs currently. But if I did incur charges, I'd see the current balance on the main page of the dashboard. In addition, I'll receive the same information in a monthly statement that is sent to the primary email account.

However, I don't recommend waiting for the bill to arrive before checking your totals. To effectively manage billing, you should check this dashboard manually, which might enable you to catch an error before the end of the month, which might save you money. The links on the left will give you additional billing information, as well as providing a way of accessing previous bills.

Adding a billing alert

In this section, I'm going to give you some high-level tips regarding some of the things you can do in order to help keep your AWS bill down.

In the previous section, I recommended that you check the bill regularly instead of only when the monthly invoice arrives. While that is good advice (if I do say so myself), the reality is that server administrators like us are busy and may not remember to check the bill on a regular basis. This is why we typically set up system alerts to notify us when our servers are encountering an issue. Similarly, we can actually set up an alert inside our AWS account that can notify us if our bill gets too high. In fact, even though the AWS account we've been working with was likely only created as a test account, it's especially important that we add a billing alert so we can be alerted if any of our experiments were misconfigured in such a way that we're incurring costs we didn't intend to.

In the *Further reading* section at the end of the chapter, I have included a link to AWS documentation that details how to set up billing alerts, and I definitely recommend you enable them. If you're using an AWS account for your organization to run actual production servers, it's a good idea to enable billing alerts there as well. In fact, be sure to create billing alerts in every AWS account you manage. Another important element to consider is removing backups that aren't needed anymore, which can also lower costs.

Removing unneeded backups

This may seem like a no-brainer, but you'd be surprised—every organization I've ever worked for or consulted with in regard to AWS has run into an issue where backups get out of control and generate large costs. My personal record for witnessing a waste of money in this manner is one company that had over 23,000 unnecessary snapshots in their account, sitting around for over 5 years. I don't remember the exact dollar amount, but this error cost them thousands of dollars a month for years!

Backups themselves are extremely important though, and the previous example might've been understandable if the organization had legal requirements that forced them to retain all backups for a specific period (such as 5 years). And backups, as you well know by now, are essential if the organization runs into some sort of issue and needs to restore something from the past. But the general rule of thumb here is to ensure that when the day comes to add an automatic backup feature, you also implement (and regularly test) an automatic cleanup procedure as well.

Running EC2 instances only when they're needed

Many organizations do business around the clock and all year round, while others conduct business only during daytime business hours. While it might not be immediately apparent why this matters, consider the fact that you don't get charged for an EC2 instance while it's powered off. You do get charged for things like storage regardless of its state, but you aren't charged for the instance itself if it is not running. If your organization has a server that is only used during certain hours, consider stopping the instance outside of that time period and then starting it back up when it's needed.

There's additional functionality in AWS that allows you to automatically schedule an instance to run only during certain hours, so there are ways to ensure your infrastructure is available when it's needed. Make sure you keep this in mind as you navigate AWS; if you're creating a new server, is it going to be necessary to make it run 24/7, or is it only necessary to run it at certain times? You'd be surprised how much money this may save you.

Stopping or terminating unneeded EC2 instances

Similar to the advice in the previous section about having instances run only when necessary, consider deleting instances completely if they're not needed at all. Unless you're utilizing a very specific type of instance, you won't be billed for an EC2 instance that doesn't exist. Make a habit of deleting things when you're done with them. If you think you may need a server again in the future but you're not sure, consider creating an AMI of the server, and then removing it. Although the AMI itself will have a cost, it's going to be less than running an actual server.

This advice isn't exclusive to EC2 instances; other services within AWS will also cause you to incur costs if you don't clean them up. Make sure to keep an eye on other components and services that charge by usage, such as RDS, S3, EBS volumes, and so on.

So, there you have it—with some basic advice, you should be able to keep your billing under control. And even if you make a mistake, as long as you've configured billing alerts, you should be notified and be able to correct the problem quickly. I can understand if some of the billing advice was overwhelming, but you'll get used to it. AWS has a lot of components that can make up a bill, but as long as you set up alerts and delete items you're not using, you shouldn't have any problems.

We've gone over quite a bit in this chapter, but where should you go from here? In the next section, I'll provide some advice for continuing your learning and taking your AWS skills to the next level.

Taking the cloud further: additional resources to grow your knowledge

AWS is a huge service, and we haven't even scratched the surface of the platform in this chapter. We've just created a simple load-balanced application in an earlier section, and we'll even learn how to automate creating cloud resources in the next chapter. But if you've found this chapter fun and want to work with AWS more and enhance your skills, I thought I'd provide some additional advice that will hopefully help you do that.

Online training and labs

There is quite a bit in the way of online resources to expand your knowledge. Some of these resources are free, such as a section of the AWS website that provides free hands-on training: <https://aws.amazon.com/training/digital/>

While you may already be aware of the value of YouTube when it comes to training videos, it's a great source of knowledge. (And you may have even stumbled across my YouTube channel, over at Learn Linux TV.)

There are many videos on YouTube that can provide training, but that's not the only source of video content; Packt Publishing features video training courses as well, in addition to Udemy, which also has some great content.

Overall, there's no shortage of training materials available, but I'd recommend starting with the free training provided by AWS that I've mentioned above, as well as the additional training content that Amazon makes available at <https://aws.training>.

Certification

While it will take a bit of work, achieving one or more certifications in AWS will lead you down a path where you'll learn the platform in much greater detail. In addition, individuals with AWS certifications are in high demand in the IT industry, so achieving certification is a good idea all around. I recommend looking into the **AWS Certified Cloud Practitioner** credential, which is a more entry-level certification that is approachable to those just starting out.

After you grow your expertise, you may want to consider the **AWS Certified Sysops Administrator** credential, which is more challenging but will boost your knowledge even further.

Keep experimenting and learning

Even if you don't decide to achieve a certification, keep experimenting with the platform. Getting your hands on the technology and making use of it on a regular basis is usually the best way to learn and keep your skills sharp. Try to create additional types of infrastructure, build and rebuild test instances, and above all, have fun. For many people, there's no greater way to learn something than to get your hands dirty and experience it. I also recommend you follow any blogs around cloud computing and related technologies as well.

AWS documentation

The documentation provided by AWS is well written and very detailed. You can learn everything you need to know from the AWS documentation alone. The documentation pages are above and beyond the typical level of quality you would expect from such a large service; Amazon takes the AWS documentation seriously. The documentation pages are available here: <https://docs.aws.amazon.com/index.html>.

New resources for learning about AWS are being created on a regular basis, so keep your eyes open for new books, training videos, and more as you study the platform. I think you'll have a lot of fun taking your skills to the next level and exploring everything the AWS platform has to offer.

Summary

This chapter has been one of the most involved in the entire book so far, and you've accomplished a lot. During the course of this chapter, you learned about AWS, set up your own cloud server, set up Auto Scaling to ensure that your server is able to automatically heal from disasters, and even set up a load balancer to enable routing between multiple instances. Make sure you take some time to let all this knowledge sink in before continuing on, and I also recommend you spend some additional time with AWS before moving on to the next chapter.

Speaking of the next chapter, we're going to work with AWS again, but this time, we're going to focus on learning Terraform, which is an awesome tool that will enable us to automate the building of our cloud resources from the ground up. It's going to be a lot of fun.

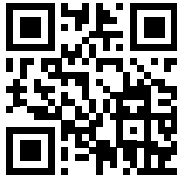
Further reading

- Amazon Elastic Compute Cloud documentation: <https://learnlinux.link/ec2-docs>
- Security groups for your VPC: <https://learnlinux.link/vpc-docs>
- Auto Scaling documentation: <https://learnlinux.link/as-docs>
- Amazon Machine Images (AMIs) : <https://learnlinux.link/ami-docs>
- Amazon Elastic Kubernetes Service documentation: <https://learnlinux.link/eks-docs>
- AWS Billing Alert documentation: <https://learnlinux.link/cw-docs>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



20

Automating Cloud Deployments with Terraform

The previous chapter was especially fun: we were able to deploy Ubuntu in the cloud, utilizing **Amazon Web Services (AWS)**. Deploying infrastructure in the cloud is very powerful and allows us to accomplish things that are not normally possible (or are very tedious) with physical infrastructure. We can spin up Ubuntu instances in minutes, and even set up auto-healing to cover us in situations that would normally result in complete service disruption.

This time around, we're going to work with cloud deployments again, and check out an awesome tool called **Terraform** that will allow us to automate the provisioning of our cloud resources. We've already explored the concept of automation back in *Chapter 15, Automating Server Configuration with Ansible*, when we learned about the basics of Ansible. Terraform allows us to take our automation to the next level and even interact with providers such as AWS directly.

In this chapter, we'll explore the following concepts:

- Why it's important to automate your infrastructure
- Introduction to Terraform and how it can fit within your workflow
- Installing Terraform
- Automating an EC2 instance deployment
- Managing security groups with Terraform
- Using Terraform to destroy unused resources
- Combining Ansible with Terraform for a full deployment solution

Why automate the building of our infrastructure? There are many benefits of doing so, and we'll take a look at some of those benefits in the next section.

Why it's important to automate your infrastructure

Automation with regards to infrastructure is an expansive topic, and it easily deserves a book of its own. In fact, there are not only books dedicated to it but entire online courses as well. There are many different utilities you can use, each with its own pros and cons. We have configuration management tools, such as Ansible, Chef, and Puppet. We looked at Ansible earlier in the book and worked through some examples to see how powerful it is. When we worked with that earlier, I'm sure you immediately saw the benefit—not having to build a solution manually is a beautiful thing.

The importance of not having to build solutions manually cannot be overstated. Perhaps the most obvious benefit is the fact that it can save you hours, or even days of work. When I first started working in IT, setting up servers was always a manual task. Sure, you could create a Bash script and automate some tasks that way, but tools specifically designed to automate will handle the task much more efficiently. An IT staff that would normally be overwhelmed at the thought of setting up a large number of servers would be able to perform the same task much quicker with automation. And with all the time that's saved, IT staff members can focus on other tasks rather than spending the majority of their time on one task.

Another benefit of automation is that the likelihood of human error is much lower. While you're building your automation solution, making mistakes is unavoidable. You may mistype something while writing a script that causes a syntax error, or perhaps something doesn't get created quite the way you expected. But after you've spent the time building your automation scripts and verified there are no errors, then you can run them again and again and the infrastructure will get created the same way each time. Compare that to having to manually set up servers each time you wish to implement a new solution, and you can imagine how often it may happen that there may be mistakes to fix. Some of which you may not even discover until later on.

Automation also has another benefit you may not expect, **Disaster Recovery**. While we will cover disaster recovery in *Chapter 23, Preventing Disasters*, it's worth mentioning now because an effective automation solution will make the process of recovery quicker. It's an administrator's worst nightmare to even think that a server that's important to your organization may someday fail, but it's a fact of life.

Our organization may have a very complex application that consists of one or more web servers, a load balancer, security settings, and more. It could take hours to rebuild a solution like that manually. But with automation, you would simply run your scripts to recreate the same solution in mere minutes. Automation itself won't protect you from losing data (which would be an even scarier problem) but at the very least it can help you to provision replacement resources quicker than if you had to do the same manually. Not only that, I presume your clients (as well as your boss) will prefer your organization's application to come back online in minutes, rather than hours or days.

In addition, your automation scripts can serve as a form of living blueprint. Even if you aren't planning on reprovisioning your servers and related infrastructure, another administrator can look at your automation scripts and understand better which components make up the overall solution, allowing them to get up to speed quicker if they're taking over the management of infrastructure from someone else.

Automation is one of those things that I probably won't have to try too hard to sell to you, because if you already have experience working in IT, then you already know how tedious it can be to manually rebuild servers.

Sometimes, it may feel as though we have more tasks to complete than we have hours in the workday. But with automation, we can get some of that time back and possibly even lower our stress level a bit. And it's not the first time we've worked with automation; we did take a look at Ansible earlier in the book, so you are probably well aware of the benefits. But what we're going to do in this chapter is implement automation at a lower level than Ansible, and we'll do so with a solution known as **Terraform**. What is Terraform, you may ask? In science, terraforming is an amazing process of taking a planet that is uninhabitable and converting it into one that is able to support life as we know it. But for our purposes, Terraform is the name of an awesome utility we can use to automate an entire cloud computing implementation. In the next section, we'll define it even more.

Introduction to Terraform and how it can fit within your workflow

Terraform is an amazing tool created by a company called Hashicorp that can automate your infrastructure at a level lower than Ansible, Puppet, or other configuration management solutions. In fact, Terraform typically doesn't replace those but complements them. With configuration management tools, we generally have to create the initial server and set up the operating system first before we can implement them. With Ansible, there are actually methods of using it to create infrastructure components, but that's beyond the scope of the book.

Not only that, but while Ansible is able to create some types of infrastructure, that's not what it does best. To understand where something like Terraform fits, it's best to think of Terraform as making things exist and Ansible as taking things that already exist and ensuring they're configured properly.

When it comes to Terraform itself, it allows you to take advantage of a neat concept, **Infrastructure as Code**. In the previous chapter, we set up an entire load-balanced application in AWS. We created an EC2 instance, as well as an AMI, and then we built the load balancer along with Auto Scaling. While that process was incredibly fun, it was a manual one. If you made mistakes during the process, you had to go and fix them. After you were done, your solution was created and working. What Terraform allows us to do is write code that represents our desired end state. When it runs, it checks the cloud provider and performs an inventory. If something we've added to our scripts isn't present with the cloud provider, it will make sure that the current state matches the desired end state in our code. We can even provision an entire cloud solution without even logging in to AWS beyond the first time.

An important consideration when it comes to automation tools is whether or not the tool is **cross-platform**. Many cloud providers feature built-in tools to do the same thing that Terraform does. For example, AWS has a feature called **CloudFormation** that allows you to script infrastructure builds, just as you can with Terraform. But the problem is that CloudFormation is specific to AWS. You can't utilize that service to build infrastructure in Microsoft Azure or Google Cloud. A tool that's cross-platform can run in any environment. We already saw this with Ansible earlier in the book: Ansible doesn't mind if the servers you're having it configure reside in AWS or even if they're physical machines in a rack. To Ansible, Ubuntu is Ubuntu, regardless of where it's running. This allows you to use the same tool in multiple environments, without having to recreate a new set of automation scripts for each one. Terraform is also a cross-platform tool.

Why does it matter if a tool is cross-platform? If you have to maintain several completely different tools that all do the same thing, it's a waste of time. If you can learn one tool and use it in every environment you support, then it's less of a maintenance burden. This is why I always recommend avoiding platform-specific tools, such as CloudFormation in AWS. There's even a tool within AWS called **OpsWorks** that's used for the same purpose as Ansible (configuration management), but again is specific to AWS.

A typical organization will pivot in different directions multiple times throughout the life of the company. An organization that is using AWS for 100% of its infrastructure may someday decide to support other cloud providers. Sometimes, all it takes is the right client or situation to make the company consider using a cloud provider for a project that would normally not be considered.

It could also be the case that a company might change primary providers due to a change made with the current platform that increases cost, or some other reason. If you use cross-platform tools, then you can take those tools with you (for the most part) if you change providers. Also, being able to support multiple providers not only makes you a more powerful administrator but also offers additional value to your organization.

Terraform itself is not going to be 100% identical between cloud platforms, though. The syntax does change from one cloud provider to another. Currently, there doesn't seem to be a solution available for Infrastructure as Code that is 100% portable between environments. But considering solutions such as CloudFormation are 0% transferable to other platforms, then Terraform still wins out in comparison since it is a tool you can use with multiple providers. The general consensus of how Terraform works will remain the same with each provider, so it's going to still save you time if you use it and then switch providers.

How does Terraform work? We'll install it in the next section and actually use it to deploy an EC2 instance in the section after that. But in a nutshell, Terraform is a utility you can download to your local laptop or desktop, and use to turn script files into actual infrastructure. It supports many different cloud platforms, such as AWS, GCP, and others. It even supports VPS providers, such as Digital Ocean and Linode. Terraform refers to each of those platforms as a provider and gives you the ability to download the appropriate plugin within Terraform to support your chosen provider(s).

As you'll see later in the chapter, Terraform allows you to test your configuration first, and preview the changes it will make. Then, if you accept the changes, it will connect to your provider and create the infrastructure as you've defined it in your code. Although we won't cover version control in this chapter, typical organizations will store their Terraform code within a Git repository or some other version control system, so that the code is safe from being accidentally deleted. In a typical organization, one or more administrators will work with the Terraform code and push their changes into the repository.

In the next section, we'll walk through the process of installing Terraform so we will have everything we need in order to get started and build some automation around our infrastructure.

Installing Terraform

The process of running Terraform and using it to provision your cloud resources is generally initiated on your local laptop or desktop. Terraform itself is downloaded from its website, and it's available for all of the leading operating systems.

Unlike the majority of applications, there's no installer. Terraform is run directly from the file you download; there's no installation process to go through. You can install it system-wide if you want to do so, but you can run it from any directory you wish. Download files for Terraform are located at the following website: <https://www.terraform.io/>.

Once there, you should see a **Download** button:

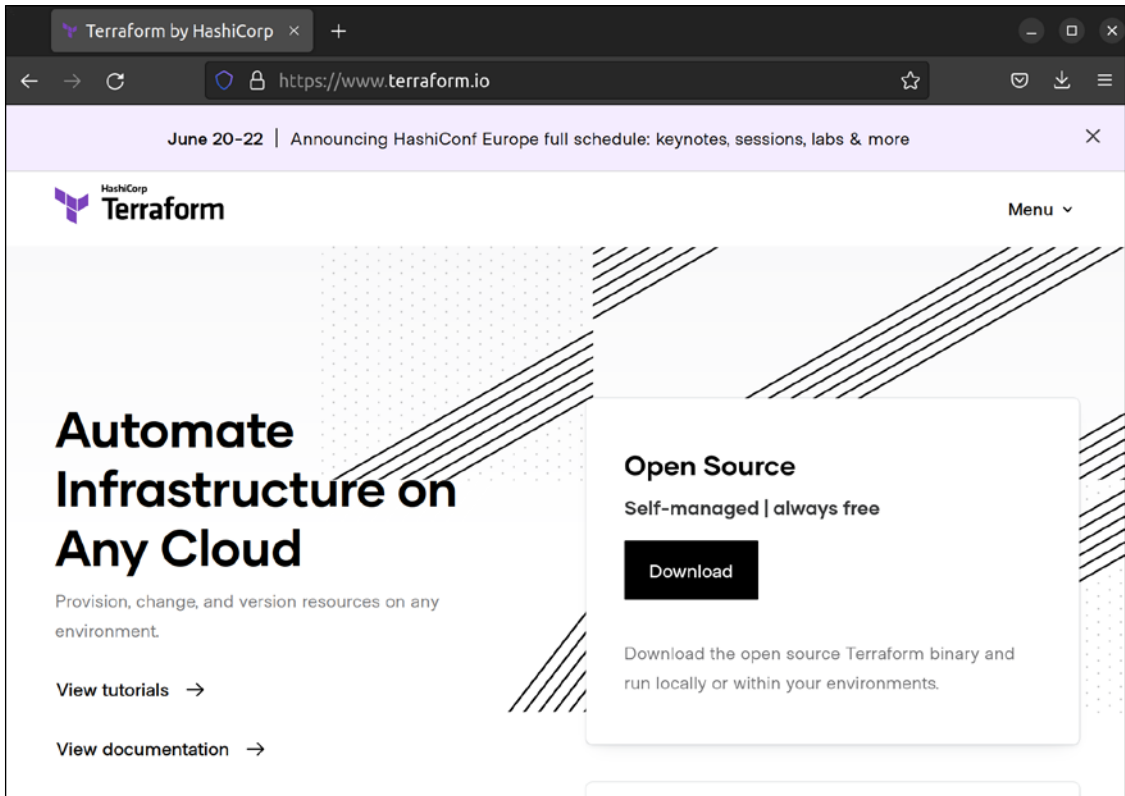


Figure 20.1: The Terraform website

After clicking the **Download** button, you'll see a new page that will offer Terraform for six different operating systems, including the usual suspects such as Linux, macOS, and Windows. Most likely, it will automatically select the operating system that the computer you're visiting the site from is using. For example, here's what the page looks like while downloading the macOS version:

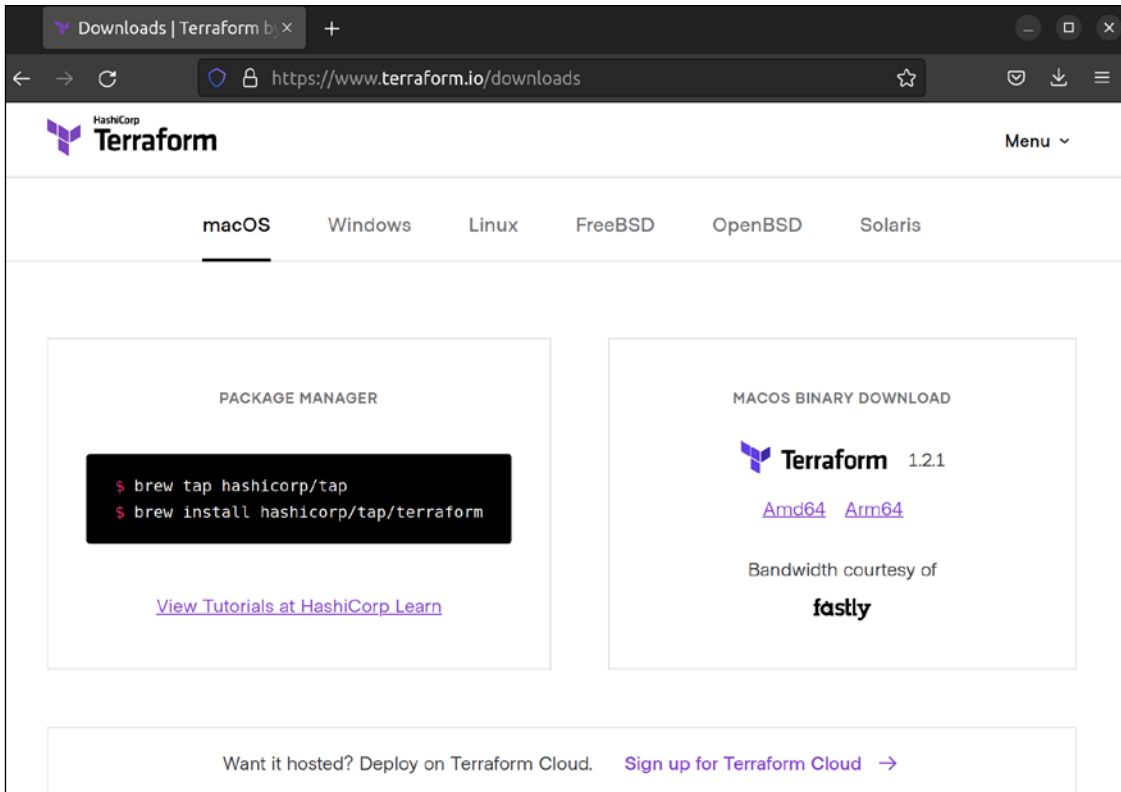
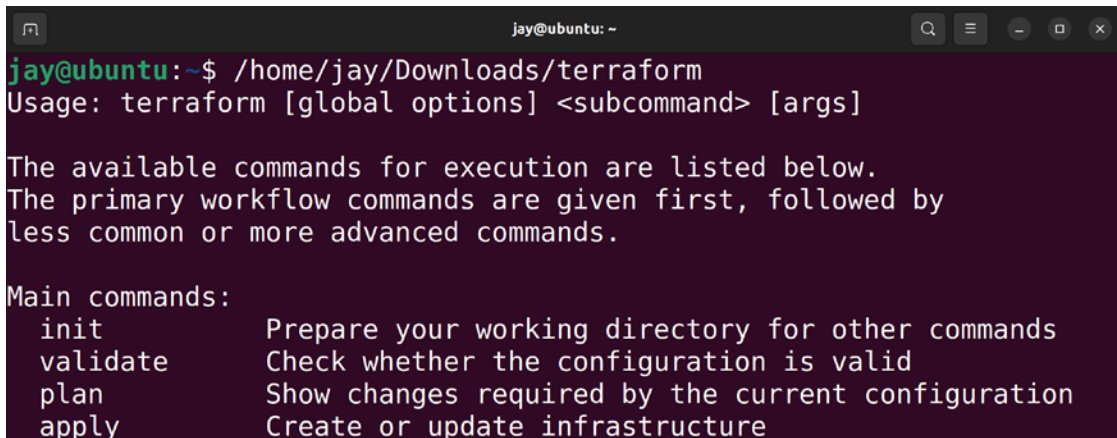


Figure 20.2: The Terraform website, downloading for macOS

At this point, all you'll have to do is download a version of Terraform specific to your operating system. Most computers sold nowadays are 64-bit, so it should be straightforward to choose which one to download. If you'd like to run Terraform from a Raspberry Pi, choose the Arm version for Linux. Once you download it, you'll have a ZIP file locally that you can extract. Inside, you'll find a binary file simply titled `terraform` and that's all you'll actually need.

You'll be able to run `terraform` right from the command prompt of your operating system:

A terminal window titled 'jay@ubuntu: ~' with search, menu, and window control icons in the title bar. The prompt is 'jay@ubuntu:~\$' followed by the command '/home/jay/Downloads/terraform'. The output shows the usage and a list of main commands: init, validate, plan, and apply, each with a brief description.

```
jay@ubuntu:~$ /home/jay/Downloads/terraform
Usage: terraform [global options] <subcommand> [args]

The available commands for execution are listed below.
The primary workflow commands are given first, followed by
less common or more advanced commands.

Main commands:
  init           Prepare your working directory for other commands
  validate       Check whether the configuration is valid
  plan           Show changes required by the current configuration
  apply          Create or update infrastructure
```

Figure 20.3: Running terraform from a terminal window with no options

In the screenshot, I typed out the entire path to the downloaded and extracted terraform file, which was saved in my home directory under the downloads folder. I ran it with no options, so it printed out the help page.

If you'd like to install it system-wide, you can move terraform into the `/usr/local/bin` directory if you're running Linux or macOS:

```
sudo mv terraform /usr/local/bin
```

The `/usr/local/bin` directory is recognized by both Linux and macOS as a directory that is searched for binary files. This concept is referred to as your `$PATH`, which is a special variable that holds all the directories your profile is set to look into when attempting to execute a command. The method of adding a new directory to your `$PATH` differs from one operating system to another, but in terms of macOS and Linux, `/usr/local/bin` is already recognized, so when you copy terraform into that directory, you should be able to simply type terraform in your terminal without needing to type the full path each time you wish to use Terraform. This is optional, but it makes it simpler.

In order for Terraform to be able to work with AWS, we'll need to generate an API key for it. This is done inside the AWS Management Console, which you should sign into now so we can create what we need. In the previous chapter, we discussed IAM, which is a service within AWS that allows you to not only create user accounts for fellow administrators but also lets you create keys for programmatic access. The latter will be how we allow Terraform to connect to our AWS account in order to perform tasks on our behalf.

Inside the IAM section of the AWS management console, click on the **Users** link that you should see in the left-hand menu, followed by the blue **Add User** button that you should see on that page. The following form will appear:

The screenshot shows the AWS IAM 'Add user' console page. The page title is 'Add user' and it shows the 'Set user details' section. The 'User name*' field contains 'terraform-provisioner'. Below it is a blue button labeled 'Add another user'. The 'Select AWS access type' section has two options: 'Access key - Programmatic access' (selected with a checked checkbox) and 'Password - AWS Management Console access' (unchecked). The 'Access key' option description states it enables an access key ID and secret access key for the AWS API, CLI, SDK, and other development tools. The 'Password' option description states it enables a password for sign-in to the AWS Management Console. At the bottom, there are 'Cancel' and 'Next: Permissions' buttons. The footer includes 'Feedback', 'Looking for language selection? Find it in the new Unified Settings', '© 2022, Amazon Web Services, Inc. or its affiliates.', 'Privacy', 'Terms', and 'Cookie preferences'.

Figure 20.4: Creating an IAM user for the purpose of running Terraform

In my case, I decided to call my user `terraform-provisioner`, but you can use whatever name you'd like. I checked the box next to **Programmatic access** and I left the second unchecked, because I do not want this user to be able to log in to the console. Click **Next: Permissions** to continue. On the next screen that comes up, we'll set the policy that the user will have access to:

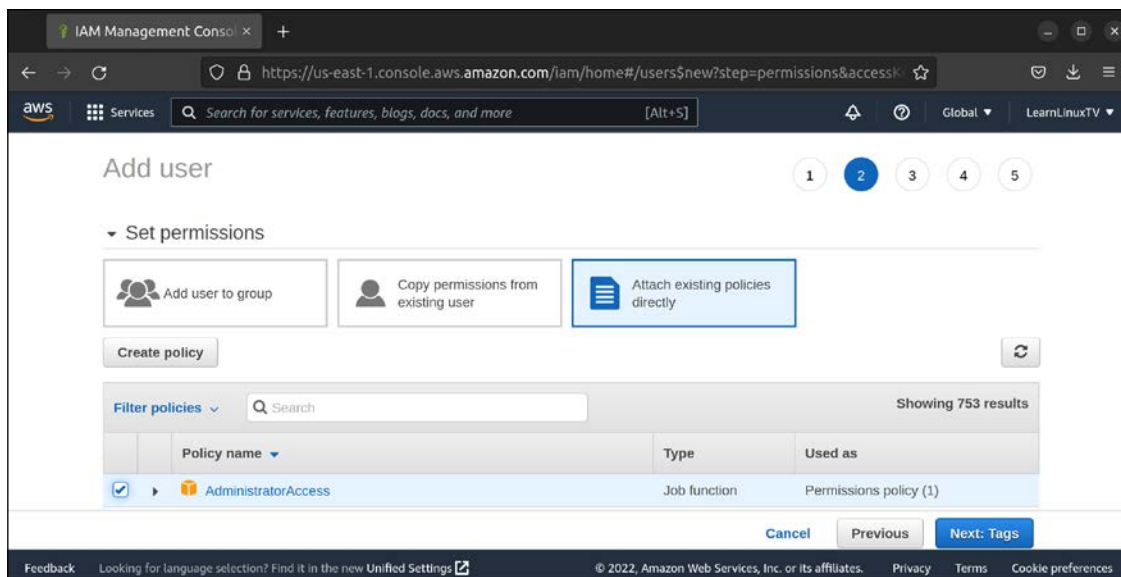


Figure 20.5: Creating an IAM user for Terraform (continued)

For this screen, click on the box that reads **Attach existing policies directly** to highlight it, and check the box below to add the **AdministratorAccess** policy to this object. This is the policy that will grant Terraform its ability to interact with AWS.

Click **Next: Tags** to continue, then on the next screen, you can skip adding tags (unless you'd like to add them) and you can click **Next: Review** and then **Create User** to finish the process.

The final screen that appears should report that the process was successful, and the **Download .csv** button gives you the ability to download your key. You can also reveal the secret access key by clicking the **Show** button, as I've done in *Figure 20.6*:

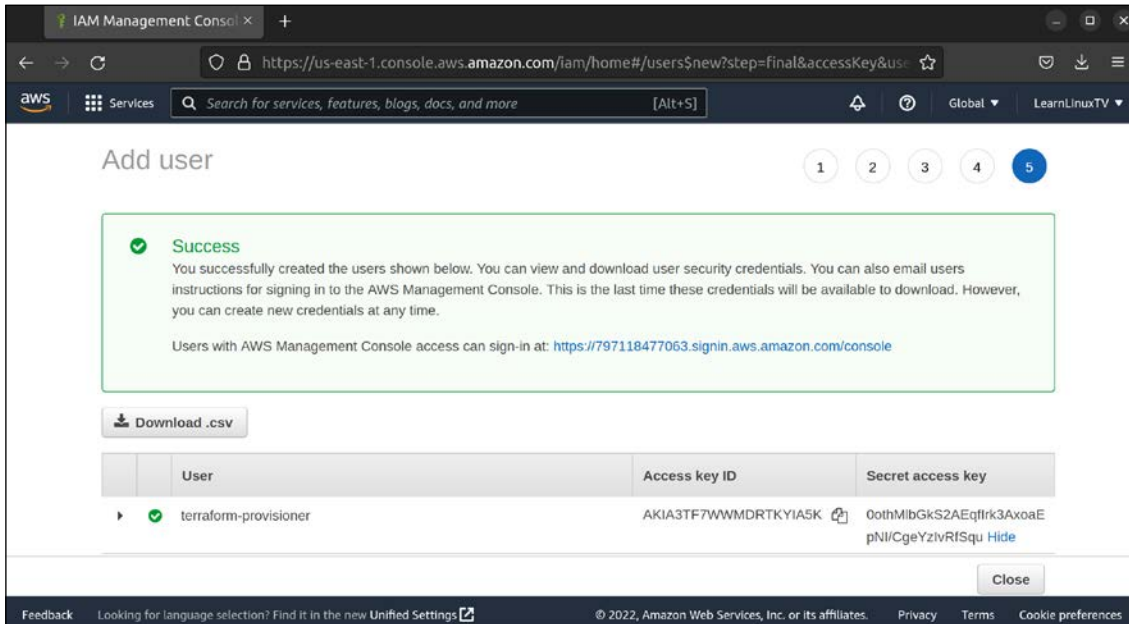


Figure 20.6: Creating an IAM user for Terraform (final screen)

I'd like to give you a few warnings about the key, though. First, whether you download the key or reveal it by clicking the **Show** button, this is the last time you'll ever see it. You won't be given another opportunity to download the full key. I recommend you download the key and store it in a safe place. You should protect the key and not let anyone have access to it, and you should definitely not upload it to a version control repository or any other resource that's publicly available. And you should absolutely not show the key in clear text in a book that a bunch of people is going to read. If this key falls into the wrong hands, then anyone that has it will be able to interact with your AWS account. Treat this key with care. The only reason I show mine here is because I want you to see what the process actually looks like. I'll delete it from my AWS account before the publishing process of this book is finalized. On your end, definitely don't let this key leak!

Now we have everything we need to proceed to build AWS resources with Terraform. In the next section, we'll create an EC2 instance.

Automating an EC2 instance deployment

Let's take a look at an example Terraform configuration file that will allow us to build an EC2 instance:

```
provider "aws" {
  region = "us-east-1"
}
```

```
resource "aws_instance" "my-server-1" {
  ami                    = "ami-09d56f8956ab235b3"
  associate_public_ip_address = "true"
  instance_type         = "t2.micro"
  key_name              = "jay_ssh"
  vpc_security_group_ids = [ "sg-0597d57383be308b0" ]
  tags = {
    Name = "Web Server 1"
  }
}
```

Terraform files are saved with a `.tf` filename extension, and as for the actual name, you can call it whatever you wish. I named mine `terraform_example_1.tf`. The underscores in the filename aren't required but make it easier to use on the command line since you won't have to escape spaces. I placed my `terraform_example_1.tf` file inside a directory of its own, which is recommended. Your Terraform configuration files should be separate from other files, so having a dedicated directory for such files is ideal.

As for the actual code itself, let's explore it section by section:

```
provider "aws" {
  region = "us-east-1"
}
```

The provider block tells Terraform what type of provider we'll be working with. We're setting that to `aws` here. As mentioned earlier, Terraform is able to work with various cloud providers, of which AWS is only one. Underneath that, we're setting the `region` variable to `us-east-1`. On your end, I recommend setting this to whatever region you were using in the previous chapter; that will make the process easier for us since we already have some resources there that we can reuse for now.

```
resource "aws_instance" "my-server-1" {
```

Here, we're starting a new resource block. Each provider has its own building blocks (resources), and specific to AWS, we can use `aws_instance`. On this line, we're also naming the instance, and calling it `my-server-1`.

Note that this is a name within Terraform we're providing, not the actual name that will be used in AWS itself. Within Terraform, we'll want to have some sort of name to refer back to this particular AWS instance if we need to refer to it again elsewhere.

```
  ami                    = "ami-09d56f8956ab235b3"
```

Next, we're choosing the AMI we'd like to use for our instance. As discussed in the previous chapter, an AMI is an image we can use to build a server in AWS. The instance ID that I used here is for the official Ubuntu 22.04 AMI that comes as default with AWS. AMIs are specific to the region they were created in, so the instance ID here is specific to `us-east-1`.

If you're also using `us-east-1`, you can use the above AMI ID as-is (so long as it's not replaced by AWS with a newer one in the future). If in doubt, you can go into the AWS console, then navigate to the EC2 console, and go through the process as if you were going to manually create an EC2 instance based on Ubuntu, and copy the AMI ID from there. Perhaps even easier, you can use the Amazon EC2 AMI Locator (provided directly by Canonical) to find an AMI ID to use: <https://cloud-images.ubuntu.com/locator/ec2/>.

You're able to filter that list by Ubuntu version as well as location. That way, you can find the AMI ID for an Ubuntu 22.04 AMI that's within your chosen region. Change the AMI ID in the example code to the one you wish to use.

Also, you'll probably notice that there are quite a few spaces in between `ami` and `= "ami-09d56f8956ab235b3"`. It's common practice with Terraform syntax to align the equal sign of every line within a block, which makes the code look cleaner. This isn't required, and nothing bad will happen if you don't line everything up perfectly, but some may argue that the overall script looks cleaner that way.

```
associate_public_ip_address = "true"
```

With this line, we're deciding to utilize a public IP address with our instance, which is required if we wish to be able to access it remotely.

```
instance_type = "t2.micro"
```

Here, we're setting the desired instance type for our newly created server. As discussed in the previous chapter, there are multiple instance types available, each with a different cost. The `t2.micro` instance type is eligible for the free tier, so that's the reason I chose it.

```
key_name = "jay_ssh"
```

In the previous chapter, when we created an EC2 instance manually, part of that process was creating an OpenSSH key. The key that you've created is registered to your AWS account, so you can refer to it by the name you gave it. I called mine `jay_ssh`, but you'll want to change this to whatever you named yours. You can see a list of your OpenSSH keys in the EC2 dashboard within AWS; there's a section in the menu called **Key Pairs** where you can remind yourself what you've named your key if you forgot.

```
vpc_security_group_ids = [ "sg-0597d57383be308b0" ]
```

During the last chapter, we created a security group that allowed both Apache and OpenSSH to communicate with our instance. When you create a security group, it's designated with its own security group ID. The security group ID I used in the example was the one that was generated for me, so it won't work for you. If you access the **Security Groups** section of the EC2 console, you can find the security group ID for the one that you've created. The ID for it should start with `sg-`, followed by a series of characters. Add yours in place of what I have for mine in the example.

```
tags = {  
  Name = "Web Server 1"  
}
```

In the last section of the example, we're setting a tag. As discussed in the previous chapter, AWS allows you to create tags that are useful information you can have attached to an instance, which can give you additional information about its intended use. The `Name` tag is a special tag that changes the name that you see for the instance in the AWS EC2 list. You can name yours whatever you'd like.

At this point, we should be all set to go ahead and run Terraform to create our instance using our Terraform file as a blueprint. First, we need to set the access and secret access keys for Terraform to use. In your terminal, you can enter the following commands to do this:

```
export AWS_ACCESS_KEY_ID="AKIAVNXBZU20BNWQQ7ET"  
export AWS_SECRET_ACCESS_KEY="KvRAFvkwUa4Vn2ZIZHGy/IKMxdMo1p1aMQoXZPVv"
```

Those commands are simply run from your terminal and create environment variables containing the required keys. Terraform will look for the existence of the `AWS_ACCESS_KEY_ID` and `AWS_SECRET_ACCESS_KEY` variables when it runs, and by exporting them, we're making them available in our session. There's actually a way you can add the keys right into the Terraform file itself, but we don't want to do that, because then the key might get uploaded somewhere public if we do upload the entire file somewhere. There's also a way to set up variables within the Terraform file to include these keys, but that's beyond the scope of this chapter.

After exporting the variables, we need to initialize Terraform to ensure it has the required components it needs to interact with AWS:

```
terraform init
```

With that example, you can add the full path to the `terraform` utility if it's not in a shared `$PATH` location, such as `/usr/local/bin`, which is a recommended location that I mentioned earlier. If you did copy it to `/usr/local/bin`, then you should be able to simply type `terraform` instead of the full path.

The `terraform init` command instructs Terraform to initialize itself. It will look at any Terraform files you have in your current working directory and look for the provider line. In our case, that's at the beginning of the file. We set it to `aws`. This will trigger Terraform to download the provider add-on for AWS:

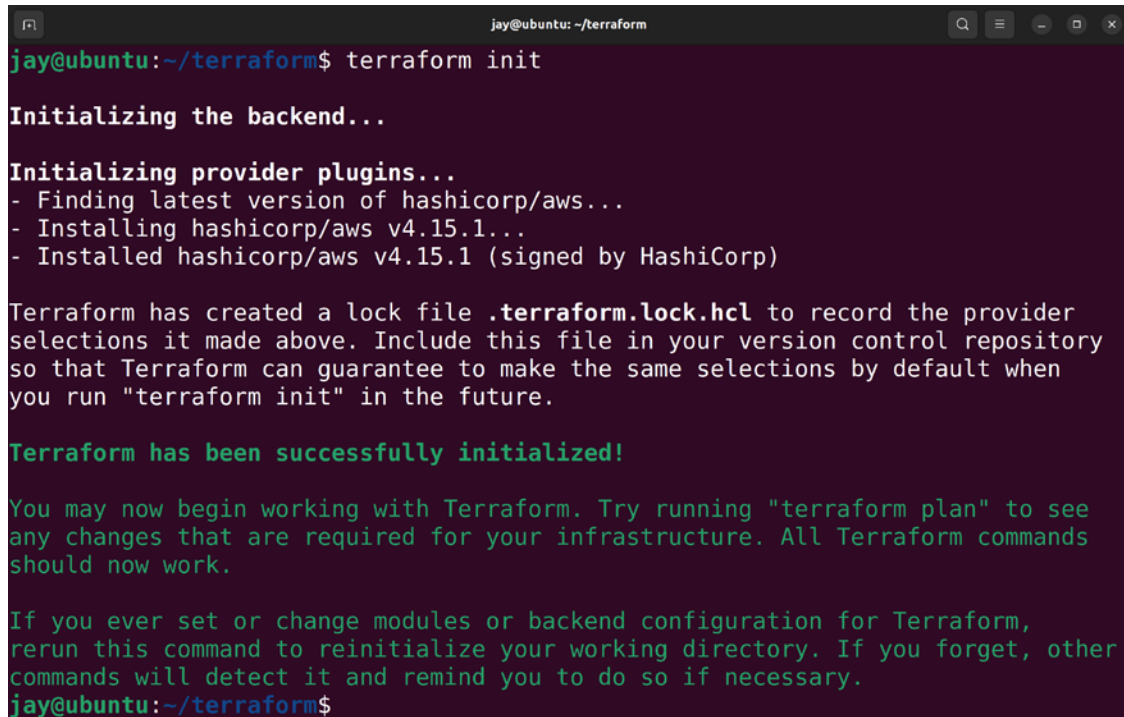
A terminal window titled 'jay@ubuntu: ~/terraform' showing the execution of 'terraform init'. The output includes: 'Initializing the backend...', 'Initializing provider plugins...', a list of actions: '- Finding latest version of hashicorp/aws...', '- Installing hashicorp/aws v4.15.1...', and '- Installed hashicorp/aws v4.15.1 (signed by HashiCorp)'. It then states: 'Terraform has created a lock file .terraform.lock.hcl to record the provider selections it made above. Include this file in your version control repository so that Terraform can guarantee to make the same selections by default when you run "terraform init" in the future.' This is followed by 'Terraform has been successfully initialized!' and instructions to run 'terraform plan' and to rerun 'init' if configuration changes. The prompt returns to 'jay@ubuntu:~/terraform\$'.

Figure 20.7: Initializing Terraform

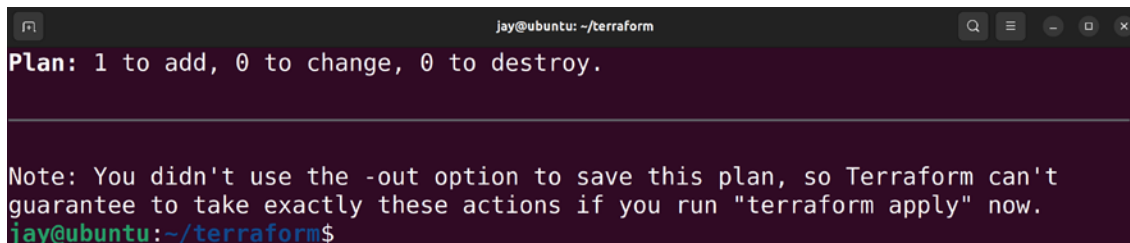
As you can see from *Figure 20.7*, when I ran the command on my end, it downloaded the AWS provider to prepare it for use.

Next, we should run what's known as a **Terraform plan**. Running a plan instructs Terraform to *not* make any changes but instead to check your syntax and ensure that you haven't mistyped anything:

```
terraform plan
```

The `terraform plan` command doesn't just check syntax, it will connect to your provider, in our case AWS, to do an inventory and compare the changes in the configuration file to the current state of the provider. If it's unable to connect to the provider, an error will be returned. If the connection is successful, Terraform will list all the changes it would've made if you instructed it to actually perform the tasks. In `plan` mode, it will never actually carry out any instructions but merely provide you with a preview.

If for some reason Terraform can't connect to your AWS account, you should make sure you've run the two `export` commands earlier, and that you've done so with the appropriate values. If you close your terminal window, you'll need to run those `export` commands again since those environment variables do not persist between terminal sessions. If successful, the Terraform plan will run:

A terminal window with a dark purple background. The title bar shows 'jay@ubuntu: ~/terraform'. The output of the 'terraform plan' command is displayed in white text. The first line is 'Plan: 1 to add, 0 to change, 0 to destroy.' Below this is a note: 'Note: You didn't use the -out option to save this plan, so Terraform can't guarantee to take exactly these actions if you run "terraform apply" now.' The prompt 'jay@ubuntu: ~/terraform\$' is visible at the bottom.

```
jay@ubuntu: ~/terraform
Plan: 1 to add, 0 to change, 0 to destroy.

Note: You didn't use the -out option to save this plan, so Terraform can't
guarantee to take exactly these actions if you run "terraform apply" now.
jay@ubuntu: ~/terraform$
```

Figure 20.8: Running a Terraform plan

In *Figure 20.8*, I've left off quite a bit of output. If your plan run was successful, Terraform will provide you with an overview of all the changes Terraform would've made if you were actually telling it to provision infrastructure.

If you would like to actually perform the changes, you can run a Terraform `apply` command. Before you do that though, always make it a habit to look at the output of the plan first. Notice the following line in the output:

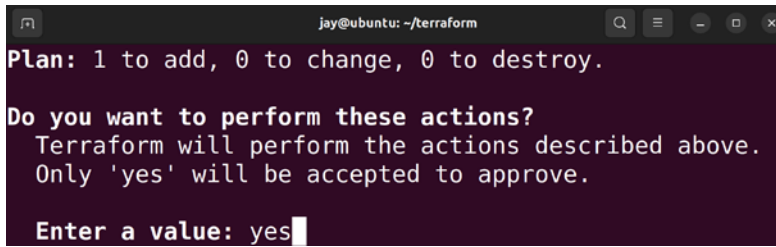
```
Plan: 1 to add, 0 to change, 0 to destroy
```

In our case, it's not going to destroy or change anything, but it's going to add something if it were to run. If you scroll up, you can find additional detail about the changes it might make if we were to run an `apply`. Pay special attention to what it might want to destroy. For some changes, Terraform may deem it necessary to delete something and recreate it from scratch. If you're using Terraform to update an existing server, you most likely won't want that server to be deleted. In that case, don't continue and run an `apply`. Always scrutinize the changes Terraform wants to make *before* you proceed and have it actually perform tasks.

Next, assuming we're comfortable with the changes, we'll proceed with an `apply`. Keep in mind that although running a plan with Terraform will cause it to look for and report syntax errors, passing the plan process with no syntax errors reported doesn't mean that there aren't any. There's only so much Terraform can do before you actually run it, so there may be errors that it can only catch during an `apply`. As you can guess, the command to run the `apply` is fairly obvious:

```
terraform apply
```

When it runs, the `terraform apply` command will run another sanity check, show the number of changes again, and ask if you'd like to continue:



```

jay@ubuntu: ~/terraform
Plan: 1 to add, 0 to change, 0 to destroy.

Do you want to perform these actions?
  Terraform will perform the actions described above.
  Only 'yes' will be accepted to approve.

Enter a value: yes

```

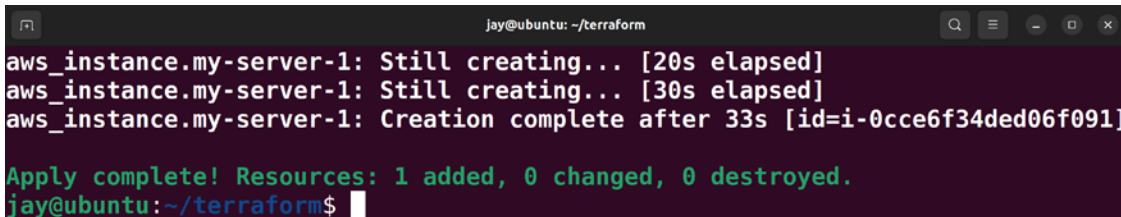
Figure 20.9: Running the `terraform apply` command

To proceed, type `yes` and press `Enter`. As it runs, you can actually see the resources you're having Terraform create show up in the AWS console as they're being provisioned. In the case of an EC2 instance as we're doing here, we'll be able to see the new instance in the list as it comes up:

<input type="checkbox"/>	Name	Instance ID	Instance state	Instance type	Status check	Alarm status	Availability Zone
<input type="checkbox"/>	Web Server 1	i-0cce6f34ded06f091	Pending	t2.micro	-	No alarms	us-east-1a

Figure 20.10: An instance showing up in the EC2 list in AWS, created by Terraform

If all goes well, Terraform itself will report the process as being successful:



```

jay@ubuntu: ~/terraform
aws_instance.my-server-1: Still creating... [20s elapsed]
aws_instance.my-server-1: Still creating... [30s elapsed]
aws_instance.my-server-1: Creation complete after 33s [id=i-0cce6f34ded06f091]

Apply complete! Resources: 1 added, 0 changed, 0 destroyed.
jay@ubuntu: ~/terraform$

```

Figure 20.11: A successful `terraform apply`

Now we've created an EC2 instance in AWS, and we did so by utilizing automation. Sure, it hasn't done all that much for us yet, but this is just a proof of concept. There are many things we can do with Terraform.

There's something missing, though—security! We should also automate the process of adding a security group to the instance, which will provide us with the access we need to be able to connect to it and manage it. We're able to access the instance now, but it's very possible that it doesn't have external internet access yet. In the next section, we'll configure the security group for the instance as well, which will allow us to configure which ports are open and which IP addresses are able to communicate with our instance.

Managing security groups with Terraform

Security groups, as you learned in the previous chapter, allow you to control what is able to communicate with your resources. In the previous section, we reused the security group that we created last time, but it would be useful to understand how to create one from scratch.

Here's the example Terraform file again, with some new code added:

```
provider "aws" {
  region = "us-east-1"
}
resource "aws_instance" "my-server-1" {
  ami                    = "ami-09d56f8956ab235b3"
  associate_public_ip_address = "true"
  instance_type          = "t2.micro"
  key_name                = "jay_ssh"
  vpc_security_group_ids = [ "${aws_security_group.external_access.
id}" ]
  tags = {
    Name = "Web Server 1"
  }
}
resource "aws_security_group" "external_access" {
  name          = "my_sg"
  description = "Allow OpenSSH and Apache"
  ingress {
    from_port = 22
    to_port   = 22
    protocol  = "tcp"
    cidr_blocks = [ "172.11.59.105/32" ]
    description = "Home Office IP"
  }
  ingress {
    from_port = 80
    to_port   = 80
    protocol  = "tcp"
    cidr_blocks = [ "172.11.59.105/32" ]
    description = "Home Office IP"
  }
  egress {
    from_port = 0
    to_port   = 0
    protocol  = "-1"
    cidr_blocks = ["0.0.0.0/0"]
  }
}
```


I've added an entirely new section to the file, but before we get to that, I also changed a line from the previous example. It's the tenth line down:

```
vpc_security_group_ids = [ "${aws_security_group.external_access.id}" ]
```

Previously, we set the security group ID for this line in the file to the security group ID that already existed, the one we created in the previous chapter. The configuration I've added further down will create a new security group, and here I'm setting the security group ID to a variable instead. The `${aws_security_group.external_access.id}` variable that was set here is known to Terraform as an **output variable**. We use an output variable for the security group ID because we have no idea what the security group ID will be since the new security group hasn't even been created yet. Therefore, we use an output variable here and reference the name of the security group we'll be creating (`external_access`) with `.id` at the end, which will reference the security group ID once it's created. That way, we can reference a security group here and assign it to the instance, without having to know ahead of time what its ID will be.

Further down the file, we begin a new section:

```
resource "aws_security_group" "external_access" {
```

With that line, we're telling Terraform we'd like to create another new resource, this time a security group. We're giving this security group a name of `external_access`, which is just a name within Terraform we can reference it as, not an actual name it will be called within AWS.

```
name = "my_sg"
```

Here, we're giving the security group its actual name, the name we'll see it shown as within AWS outside of Terraform.

```
description = "Allow OpenSSH and Apache"
```

For the description line, there's nothing too surprising here: we're giving it a description we can use to describe its purpose and what the security group will be used for. Similar to the security group we've created manually in the previous chapter, we'll be opening up OpenSSH and Apache with this security group.

```
ingress {
  from_port = 22
  to_port   = 22
  protocol  = "tcp"
  cidr_blocks = [ "172.11.59.105/32" ]
  description = "Home Office IP"
}
```

The `ingress` block allows us to set a port to allow connections to come in from; in this case, we're allowing connections from port 22, which, as you probably already know, is the default port for OpenSSH. We don't want to open this port up to receive connections from the entire public internet, so we're allowing the incoming traffic to this port only if it's coming from an IP address of `172.11.59.105/32`.

In your case, you can replace that with the public IP address of your home office or organization.

The second ingress block is the same as the first, only this time it's allowing connections to port 80 for Apache:

```
ingress {
  from_port = 80
  to_port   = 80
  protocol  = "tcp"
  cidr_blocks = [ "172.11.59.105/32" ]
  description = "Home Office IP"
}
```

We also add an egress security group rule as well because, without this, our instance will not be able to reach the internet:

```
egress {
  from_port = 0
  to_port   = 0
  protocol  = "-1"
  cidr_blocks = ["0.0.0.0/0"]
}
```

As with the previous example, we'll need to run a plan and then an apply to transform our new code into reality. I'll leave it up to you to run both; as long as you haven't mistyped anything, it should apply the changes and add the new security group. Inside AWS, you should see the new security group in the console, and also see it applied to your EC2 instance. Unless you reused the AMI from the previous chapter with Apache built in, you won't be able to connect to the instance via port 80 since Apache is probably not installed, but I added it just to show you an example.

At this point, I recommend that you play around with the Terraform script we have so far, to get a feel for its syntax. Feel free to implement something extra; you can refer to the Terraform documentation for additional resources you can create with Terraform.

Congratulations on using Terraform to provision your infrastructure. Now, let's use Terraform to destroy stuff.

Using Terraform to destroy unused resources

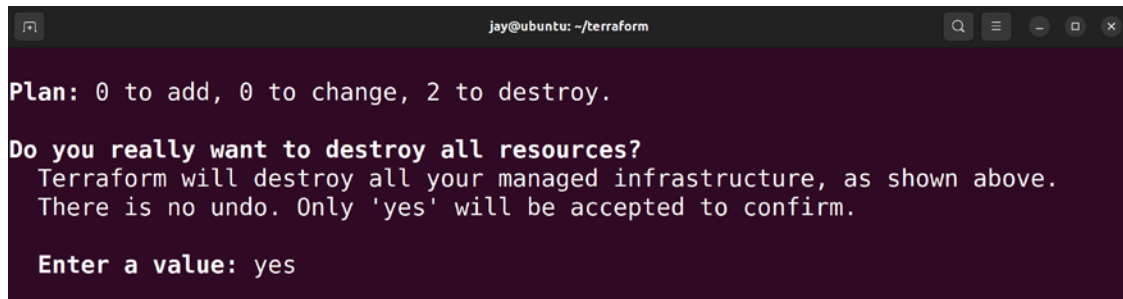
Although Terraform's primary purpose is to create infrastructure, it can also be used to delete infrastructure as well. This function is known as a **Terraform destroy**. With `destroy`, Terraform will attempt to remove all infrastructure that's defined in your configuration file. At this point, our configuration file creates an EC2 instance, as well as a security group. If we run `destroy` against it, then both resources will be removed.

Removing infrastructure with Terraform will likely be a use case you won't utilize as often as creating resources. One of the values of the `destroy` functionality, though, is that you can use it to “reset” a test environment, by removing everything defined in the file. Then you're free to use the same script to create everything again. On my end, I learn a lot faster by breaking things and fixing them repeatedly. You really shouldn't run a `destroy` job against production infrastructure that you care about, but if you're just using Terraform in a test account that doesn't have any important instances inside it, then you can continually build and dismantle your test resources over and over as you learn. Another benefit is that an organization may test a Terraform build for a client in a test account first before implementing it in production, and you can verify that everything will be built correctly before performing the actual work for the client.

Performing a `destroy` within Terraform is just as simple as previous examples:

```
terraform destroy
```

Just like before, we'll get confirmation first before it removes everything, showing us exactly what Terraform wants to remove when a `destroy` task is run:



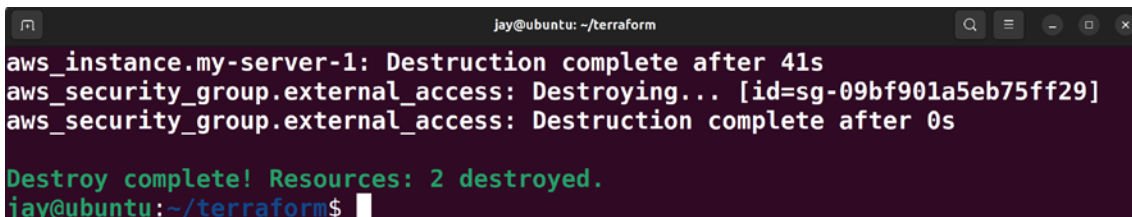
```
jay@ubuntu: ~/terraform
Plan: 0 to add, 0 to change, 2 to destroy.

Do you really want to destroy all resources?
Terraform will destroy all your managed infrastructure, as shown above.
There is no undo. Only 'yes' will be accepted to confirm.

Enter a value: yes
```

Figure 20.12: Preparing to run `terraform destroy` to remove resources

Pay careful attention to what Terraform wants to delete when you run it with the `destroy` option. The screenshot doesn't show the full output; it's quite long. Similar to `apply`, if you scroll up, you'll see that the output will contain detail about what in particular will be removed if we agree to continue. If you type `yes` and press `Enter`, the resources identified will be destroyed, and you'll receive a confirmation message confirming that the task was carried out:



```
jay@ubuntu: ~/terraform
aws_instance.my-server-1: Destruction complete after 41s
aws_security_group.external_access: Destroying... [id=sg-09bf901a5eb75ff29]
aws_security_group.external_access: Destruction complete after 0s

Destroy complete! Resources: 2 destroyed.
jay@ubuntu: ~/terraform$
```

Figure 20.13: Final confirmation after destroying previously provisioned resources

Basic usage of the `terraform` command is logically structured; we looked at how to run a `plan` as well as an `apply`, and now we know how to destroy our resources as well so we can start over with a clean slate. The majority of the time spent learning Terraform will be a matter of learning the syntax of its config files, but that will come in time. At this point in our journey, we should have a solid foundation we can build upon.

However, we're not done yet! I've referenced Ansible several times in this chapter, reminding you about the fact that we used it in the past to configure a server. But what if I told you we can combine Terraform and Ansible? We certainly can, and we'll do so in the next section.

Combining Ansible with Terraform for a full deployment solution

One of the best things about automation tools is that they can often be combined to offer a shared benefit. Ansible is one of my favorite tools: you can automate the installation of packages, the creation of users, the copying of files, or most other tasks you can think of. If you are able to perform a task on the command line, chances are Ansible can automate it. Terraform, as you just saw, is really good at creating new infrastructure and automating the initial setup of servers, as well as networks and settings for AWS and other platforms. If we combine the two, it gets even better.

I find the duo of Terraform and Ansible to be a great fit. Combining these two solutions works well in my experience; we can use Terraform to create our initial server and infrastructure builds, and then use Ansible to automate future enhancements. But it's actually even better than that; we can configure Terraform to actually launch the initial Ansible run for us, so we only have to run a single script. After Terraform creates the infrastructure, provisioning of additional settings is handed off to Ansible. It's a great combination.

How does it work? In the previous chapter, we explored the concept of user data, which is a feature within AWS that allows you to run a script as an instance is being created. We used it to install all the patches and then proceed and install Apache. The example we went over was a simple Bash script and wasn't very exciting in and of itself. Sure, it did work, but we can implement a better solution. And you know what? We already have. In *Chapter 15, Automating Server Configuration with Ansible*, we were able to utilize Ansible Pull, a special mode of Ansible that allows us to pull code from a repository, and run it locally on our instance. The Ansible playbook we wrote installs Apache for us, the same as our Bash script did in the previous chapter. As a refresher, we are able to run the following command to trigger Ansible Pull:

```
ansible-pull -U https://github.com/myusername/ansible.git
```

Of course, this requires Ansible itself to be installed, and the repository needs to already exist. If you have already followed along in that chapter and you still have the repository we've created, then you already have what you need to combine Ansible with Terraform.

To save you the trouble of flipping back to *Chapter 15, Automating Server Configuration with Ansible*, here's the final `local.yml` file we ended up with:

```

---
- hosts: localhost
  become: true
  tasks:
    - name: Install Apache
      apt: name=apache2
    - name: Start the apache2 services
      service:
        name: apache2
        state: started
    - name: Copy index.html
      copy:
        src: index.html
        dest: /var/www/html/index.html

```

As you can see, this playbook is installing Apache, starting it, and also copying an `index.html` file to replace the default web page. It's fairly easy to implement this in Terraform. Here's our Terraform script again, with a new line added, shown in bold:

```

provider "aws" {
  region = "us-east-1"
}

resource "aws_instance" "my-server-1" {
  ami = "ami-0dba2cb6798deb6d8"
  associate_public_ip_address = "true"
  instance_type = "t2.micro"
  key_name = "jay_ssh"
  vpc_security_group_ids = [ "${aws_security_group.external_access.
id}" ]
  user_data = file("bootstrap.sh")
  tags = {
    Name = "Web Server 1"
  }
}

resource "aws_security_group" "external_access" {
  name = "my_sg"
  description = "Allow OpenSSH and Apache"
  ingress {
    from_port = 22
    to_port = 22

```

```

    protocol    = "tcp"
    cidr_blocks = [ "173.10.59.105/32" ]
    description = "Home Office IP"
  }
  ingress {
    from_port = 80
    to_port   = 80
    protocol  = "tcp"
    cidr_blocks = [ "173.10.59.105/32" ]
    description = "Home Office IP"
  }
  egress {
    from_port = 0
    to_port   = 0
    protocol  = "-1"
    cidr_blocks = ["0.0.0.0/0"]
  }
}

```

The new addition to the file is on line #11. We're referencing a bootstrap script, and in that script, we'll add any commands we wish to run on the newly created instance:

```
user_data = file("bootstrap.sh")
```

`bootstrap.sh` will need to exist in the same directory as the Terraform configuration file itself. The file doesn't exist yet though, so go ahead and create it, and inside you can place the following lines:

```
#!/bin/bash
sudo apt update
sudo apt install -y ansible
sudo ansible-pull -U https://github.com/myusername/ansible.git
```

We haven't made an overly complex change to the file, but what we did add gives us a great deal of benefits. The `user_data` option allows us to leverage the same user data function that's built into AWS and schedule commands to run when an instance is first created. In this example, we utilize the `user_data` option to run a series of commands against the new instance, which will install Ansible and then launch `ansible-pull` to download a repository containing an Ansible playbook and run it locally. The playbook itself was set up in *Chapter 15, Automating Server Configuration with Ansible*, so we're just leveraging what we've already created in the past, and we're having Terraform kick off the Ansible job for us when it brings up the instance.

That brings us to the end of this chapter. I hope setting up automation with Terraform was a fun experience; I definitely enjoy working with it.

Summary

There are many configuration management and provisioning tools available for automating our infrastructure builds. In this chapter, we took a look at Terraform, and then we even combined it with Ansible, which we were already using. Using Terraform, we were able to automate the creation of an EC2 instance in AWS, along with a security group to control how it can be accessed. Terraform is a very large subject, and the concepts contained in this chapter are only the beginning. There's so much more you can do with Terraform, and I highly recommend you keep practicing with it and coming up to speed.

In the next chapter, we're going to learn some methods we can utilize to add additional security to our Ubuntu servers. While no server is bulletproof, there's a basic level of security we can implement that will make it less likely for our server to be compromised. It will be a very important chapter, so you won't want to miss it.

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaz0>



21

Securing Your Server

It seems like every month there are new reports about companies getting their servers compromised. In some cases, entire databases end up freely available on the internet, which may even include sensitive user information that can aid miscreants in stealing identities. Linux is a very secure platform, but it's only as secure as the administrator who sets it up. Security patches are made available on a regular basis, but they offer no value unless you install them. OpenSSH is indispensable for remote administration, but it's also a popular target for threat actors trying to break into servers. Backups are a must-have but are potentially useless if they're not tested regularly or they fall into the wrong hands. In some cases, even your own employees can cause intentional or unintentional damage. In this chapter, we'll look at some of the ways you can secure your servers from threats.

In this chapter, we will cover:

- Lowering your attack surface
- Understanding and responding to **Common Vulnerabilities and Exposures (CVEs)**
- Installing security updates
- Automatically installing patches with the Canonical Livepatch service
- Securing OpenSSH
- Installing and configuring Fail2ban
- MariaDB best practices for secure database servers
- Setting up a firewall
- Encrypting and decrypting disks with **Linux Unified Key Setup (LUKS)**
- Locking down sudo

To get started, let's first talk about ways you may be able to lower your attack surface.

Lowering your attack surface

Your Ubuntu Server installations will likely have one or more important applications running on them, some of which might be available to the public internet. This is very common for web servers, for example, as it's the primary goal of a web server to offer a website that your users can access.

Every application that is accessible from outside the walls of your organization is a potential entry point for threat actors who might attempt to break into your server. The **attack surface** of a server is essentially a list of all the things that are potentially exploitable. In regards to security, it's important to understand which applications must be accessible remotely, and which ones you can lock down. Every application you lock down lowers the likelihood of it being taken over by an outside threat. The process of locking things down is what we refer to as lowering your attack surface.

Ideally, in a perfect world, we would disallow all outside connections to all of our servers. Threat actors can't break into a server that is completely inaccessible from the outside. That doesn't mean that there aren't any threats at all, as disgruntled employees are always a potential risk. But a server that's completely inaccessible is the most secure of all. However, it's often not feasible to disallow all outside connections. If your company provides a popular public website, then it has to be publicly available. However, if you have an application running on your server that is only used by users internally, then you should lock it down if you can. Whenever possible, it's good to implement a policy that outside connections are always disallowed by default unless there's a business need to open it up.

What do we mean by "disallow?" There are multiple ways you can disallow access to an application on your server. The most effective of these is to completely uninstall the application. If you don't have an application installed at all, it's impossible for it to be a problem. It probably goes without saying that you should uninstall applications that aren't necessary, but the entire point of running a server is to serve resources to users, so you'll always have applications running on your server (otherwise there, wouldn't be a point in having a server at all to begin with). Aside from removing an application, you can utilize a firewall to only allow specific connections. We'll actually take a look at setting up a firewall later on in this chapter.

Most importantly, after a new server is implemented, an administrator should always perform a security check to ensure that it's as secure as it can possibly be. No administrator can think of everything, and even the best among us can make a mistake, but it's always important that we do our best to ensure we secure a server as much as we can. There are many ways you can secure a server, but the first thing you should do is lower your attack surface. This means that you should close as many holes as you can, and limit the number of things that outsiders could potentially be able to access. In a nutshell, if it's not required to be available from the outside, lock it down. If it's not necessary at all, remove it.

To start inspecting your attack surface, the first thing you should do is see which ports are listening for network connections. When an attacker wants to break into your server, it's almost certain that a port scan is the first thing they will perform. They'll inventory which ports on your server are listening for connections, determine what kind of application is listening on those ports, and then try a list of known vulnerabilities to try to gain access. To inspect which ports are listening on your server, you can do a simple port query with the `ss` command:

```
sudo ss -tulpn
```

The `sudo` portion of that command is optional, but if you do include `sudo`, you'll see more information in the output. Normally I'd include a screenshot here, but there's so much information that it won't fit on this page. From the output, you'll see a list of ports that are listening for connections. If the port is listening on `0.0.0.0`, then it's listening for connections from any network.

This is potentially bad. If the port is listening on `127.0.0.1`, then it's not actually accepting outside connections. Take a minute to inspect one of your servers with the `ss` command and note which services are listening for outside connections.

Armed with the knowledge of what ports your server is listening on, you can make a decision about what to do with each one. Some of those ports may be required, as the entire purpose of a server is to serve something, which usually means communicating over the network. All of these legitimate ports should be protected in some way, which usually means configuring the service after reviewing its documentation for best practices (which will depend on the particular service) or enabling a firewall, which we'll get to in the *Setting up a firewall* section. If any of the ports are not needed, you should close them down. You can either stop their daemon and disable it, or remove the package outright. I usually go for the latter, since it would just be a matter of reinstalling the package if I changed my mind.

OpenSSH is a service that you're almost always going to have running on your servers. As you are already well aware, it's a great tool for remote administration. But as useful as it is, it's usually going to be the first target for any attacker attempting to gain entry into your server.

We won't want to remove this though, because it's something we'll want to take advantage of. What should we do? Not to worry, I'll be dedicating a section to securing OpenSSH later in this chapter. I mention this now in order to make sure you're aware that lowering your attack surface will absolutely need to include at least a basic amount of security tweaking for OpenSSH. In addition, I'll go over Fail2ban in this chapter as well, which can help add an additional layer of security to OpenSSH.

As I've mentioned, I'm a big fan of removing packages that aren't needed. The more packages you have installed, the larger your attack surface is. It's important to remove anything you don't absolutely need. Even if a package isn't listed as an open port, it could still be leveraged in a **vulnerability chain**. If an attacker uses a vulnerability chain, that essentially means that they first break into one service and then use a vulnerability in another (possibly unrelated) package to elevate their privileges and attempt to gain full access. For that reason, I will need to underscore the fact that you should remove any packages you don't need on your server. An easy way to get a list of all the packages you have installed is with the following command:

```
dpkg --get-selections > installed_packages.txt
```

This command will result in the creation of a text file that will contain a list of all the packages that you have installed on your server. Take a moment to look at it. Does anything stand out that you know for sure you don't need? You most likely won't know the purpose of every single package, and there could be hundreds or more. A lot of the packages that will be contained in the text file are distribution-required packages you can't remove if you want your server to boot up the next time you go to restart it. If you don't know whether or not a package can be removed, do some research on Google. If you still don't know, maybe you should leave that package alone and move on to inspect others. By going through this exercise on your servers, you'll never really remember the purpose of every single package and library, but you should still find some things that you'll be able to clean up. Eventually, you'll come up with a list of typical packages most of your servers don't need, which you can make sure are removed each time you set up a new server. You could even curate a list of unneeded packages, and then create an Ansible playbook to make sure they're not installed.

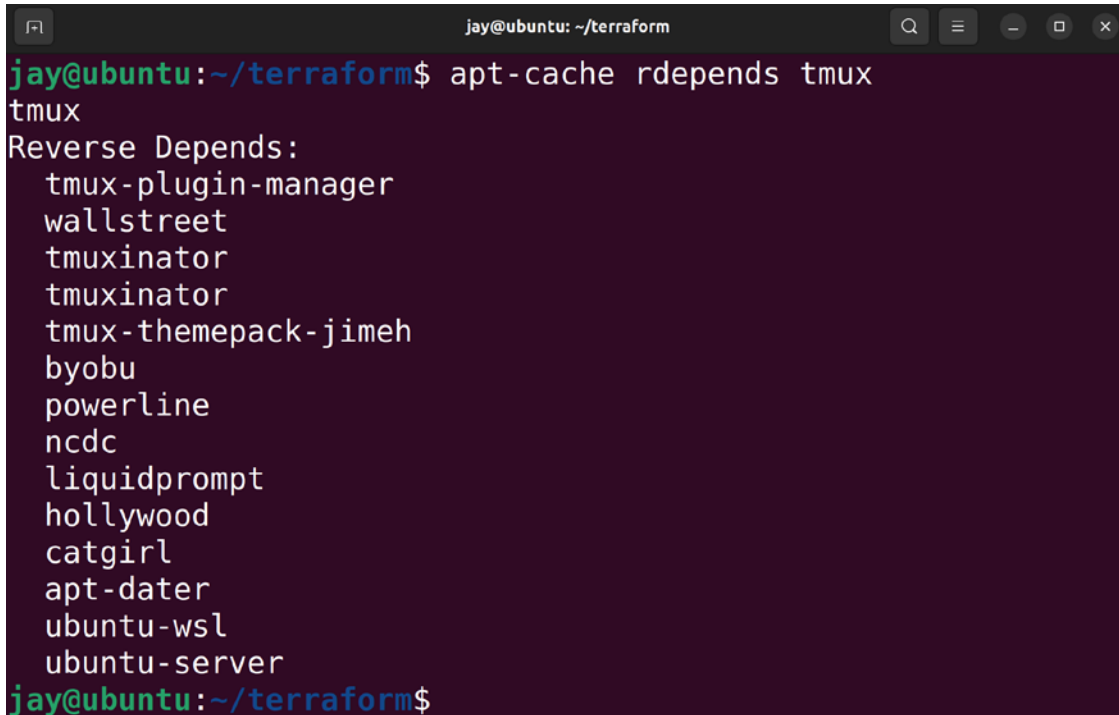
While attempting to clean up unneeded packages, a useful trick is to use the following command to check whether or not other packages depend on the package you are thinking of removing:

```
apt-cache rdepends <package-name>
```

As an example, I ran that command against the `tmux` package that I installed on a test server, but you can use whichever package name you'd like as an argument to check to see if anything depends on it:

```
apt-cache rdepends tmux
```

The output I received on my end is the following:

A terminal window screenshot showing the command `apt-cache rdepends tmux` and its output. The terminal title is `jay@ubuntu: ~/terraform`. The output lists the package `tmux` and its reverse dependencies: `tmux-plugin-manager`, `wallstreet`, `tmuxinator` (listed twice), `tmux-themepack-jimeh`, `byobu`, `powerline`, `ncdc`, `liquidprompt`, `hollywood`, `catgirl`, `apt-dater`, `ubuntu-wsl`, and `ubuntu-server`. The prompt `jay@ubuntu:~/terraform$` is visible at the end of the output.

```
jay@ubuntu:~/terraform$ apt-cache rdepends tmux
tmux
Reverse Depends:
  tmux-plugin-manager
  wallstreet
  tmuxinator
  tmuxinator
  tmux-themepack-jimeh
  byobu
  powerline
  ncdc
  liquidprompt
  hollywood
  catgirl
  apt-dater
  ubuntu-wsl
  ubuntu-server
jay@ubuntu:~/terraform$
```

Figure 21.1: Updating packages on an Ubuntu server

With the output of the previous command, you can easily identify if another package depends on the package you are thinking about removing. In the example output, we can see that `tmux` is actually installed as a dependency of the `ubuntu-server` package. This means that `tmux` is quite possibly installed by default on your server, but that may vary depending on whether or not you've installed Ubuntu Server yourself or are using a cloud image. Cloud providers don't always configure Ubuntu Server images the same way. But at the very least, you can identify dependencies and make a more informed decision on whether or not you can safely remove a package.



Even if the output shows a package has no dependencies, you still may not want to remove it unless you understand the functionality it provides and what impact removing the package may have on your system. You can always Google the package name for more details, but at the very least you should look for open ports and focus on those first, since open ports have a greater impact on the security of your server. We'll look at this in more detail later in this chapter, in the *Setting up a firewall* section.

Another important consideration is making sure to use only strong passwords. This probably goes without saying, since I'm sure you already understand the importance of strong passwords. However, I've seen hacks recently in the news caused by administrators who set weak passwords for their external-facing database or web console, so you never know. The most important rule is that if you must have a service listening for outside connections, then it absolutely must have a strong, randomly generated password. Granted, some daemons don't have a password associated with them (Apache is one example; it doesn't require authentication for someone to view a web page on port 80). However, if a daemon does have authentication, it should have a very strong password. OpenSSH is an example of this. If you must allow external access to OpenSSH, that user account should have a strong randomly generated password. Otherwise, it will likely be taken over within a couple of weeks by a multitude of bots that routinely go around scanning for these types of things. In fact, it's best to disable password authentication in OpenSSH entirely, which we will do later in this chapter. Disabling password authentication increases the security around OpenSSH quite a bit.

Finally, it's important to employ the **principle of least privilege** for all your user accounts. You've probably gotten the impression from several points I've made throughout the book that I distrust users. While I always want to think the best of everyone, sometimes the biggest threats can come from within (disgruntled employees, accidental deletions of critical files, and so on). Therefore, it's important to lock down user accounts as much as possible, and allow them access to only what they actually need to perform their job. This may involve, but is certainly not limited to:

- Adding a user to the smallest possible number of groups
- Defaulting all network shares to read-only (users can't delete what they don't have permission to delete)
- Routinely auditing all your servers for user accounts that have not been logged into for a time
- Setting account expirations for user accounts, and requiring users to reapply to maintain account status (this prevents hanging user accounts)
- Allowing user accounts to access as few system directories as possible (preferably none, if you can help it)
- Restricting sudo to specific commands (more on that later on in this chapter)

Above all, make sure you document each of the changes you make to your servers, in the name of security. After you develop a good list, you can turn that list into a security checklist to serve as a baseline for securing your servers. Then, you can set reminders to routinely scan your servers for unused user accounts, unnecessary group memberships, and any newly opened ports.

Now you should have some good ideas on how you can lower your attack surface. It's also important to keep up to date on the current trends and notices surrounding security issues that were reported. In the next section, we'll take a look at **Common Vulnerabilities and Exposures (CVEs)**, which can help you better understand the nature of threats in the wild.

Understanding and responding to CVEs

I've already mentioned some of the things you can do in order to protect your server from some common threats, and I'll give you more tips later on in this chapter. But how does one know when there's a vulnerability that needs to be patched? How do you know when to take action? The best practices I'll mention in this chapter will only go so far; at some point, there may be some sort of security issue that will require you to do something beyond generating a strong password or locking down a port.

The most important thing to do is to keep up with the news. Subscribe to sites that report news on security vulnerabilities, and I'll even place a few of these in the *Further reading* section of this chapter. When a security flaw is revealed, it's typically reported on these sites and given a CVE number, where security researchers will document their findings.

CVEs are found in special online catalogs detailing security vulnerabilities and their related information. In fact, many Linux distributions (Ubuntu included) maintain their own CVE catalogs with vulnerabilities specific to their platform. On such a page, you can see which CVEs the version of your distribution is vulnerable to, which have been responded to, and what updates to install in order to address them.

Often, when a security vulnerability is discovered, it will receive a CVE identification right away, even before mitigation techniques are known. In my case, I'll often watch a CVE page for a flaw when one is discovered, and look for it to be updated with information on how to mitigate it once that's determined.

Most often, closing the hole will involve installing a security update, which the security team for Ubuntu will create to address the flaw. In some cases, the new update will require restarting the server or at least a running service, which means I may have to wait for a maintenance period to perform the mitigation.

I recommend taking a look at the Ubuntu CVE tracker, available at <https://ubuntu.com/security/cves>. On this site, Canonical (the makers of Ubuntu) keeps information regarding CVEs that affect the Ubuntu platform. There, you can get a list of vulnerabilities that are known to the platform as well as the steps required to address them. There's no one rule about securing your server, but paying attention to CVEs is a good place to start. We'll go over installing security updates in the next section, which is the most common method of mitigation.

Installing security updates

Since I've mentioned updating packages several times, let's have a formal conversation about it. Updated packages are made available for Ubuntu quite often, sometimes even daily. These updates mainly include the latest security updates but may also include new features. Since Ubuntu 22.04 is an LTS release, security updates are much more common than feature updates.

Installing the latest updates on your server is a very important practice, but, unfortunately, it's not something that all administrators keep up with for various reasons.

When installed, security updates very rarely make many changes to your server, other than helping to keep it secure against the latest threats. However, it's always possible that a security update that's intended to fix a security issue ends up breaking something else. This is rare, but I've seen it happen. When it comes to production servers, it's often difficult to keep them updated, since it may be catastrophic to your organization to introduce change within a server that's responsible for a large portion of your profits. If a server goes down, it could be very costly. Then again, if your servers become compromised and your organization ends up the subject of a CNN hacking story, you'll definitely wish you had kept your packages up to date!

The key to a happy data center is to test all updates before you install them. Many administrators will feature a system where updates will *graduate* from one environment to the next. For example, some may create virtual clones of their production servers, update them, and then see whether anything breaks. If nothing breaks, then those updates will be allowed on the production servers.

In a clustered environment, an administrator may just update one of the production servers, see how it gets impacted, and then schedule a time to update the rest. In the case of workstations, I've seen policies where select users are chosen for security updates before they are uploaded to the rest of the population. I'm not necessarily suggesting you treat your users as guinea pigs, but everyone's organization is different, and finding the right balance for installing updates is very important. Although these updates represent change, there's a reason that Ubuntu's developers went through the hassle of making them available. These updates fix issues, some of which are security concerns that are already being exploited as you read this.

To begin the process of installing security updates, the first step is to update your local repository index. As we've discussed before, the way to do so is to run `sudo apt update`. This will instruct your server to check all of its subscribed repositories to see whether any new packages were added or whether any out-of-date packages were removed. Then, you can start the actual process.

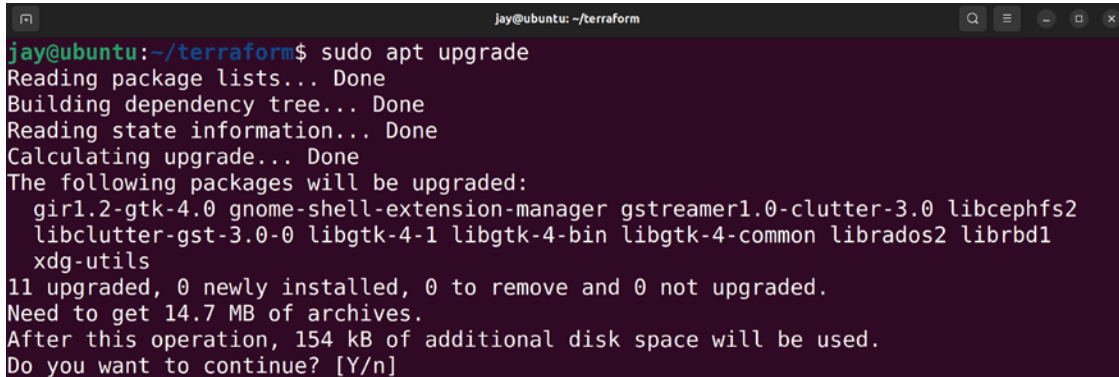
There are two commands you can use to update packages. You can run either `sudo apt upgrade` or `sudo apt dist-upgrade`.

The difference is that running `apt upgrade` will not remove any packages and is the safest to use. However, this command won't pull down any new dependencies either. Basically, the `apt upgrade` command simply updates any packages on your server that have already been installed, without adding or removing anything. Since this command won't install anything new, this also means your server will not have updated kernels installed either.

The `apt dist-upgrade` command will update absolutely everything available. It will make sure all packages on your server are updated, even if that means installing a new package as a dependency that wasn't required before. If a package needs to be removed in order to satisfy a dependency, it will do that as well. If an updated kernel is available, it will be installed. If you use this command, just take a moment to look at the proposed changes before you agree to have it run, as it will allow you to confirm the changes during the process.

Generally speaking, the `dist-upgrade` variation should represent your end goal, but it's not necessarily where you should start. Updated kernels are important, since your distribution's kernel receives security updates just like any other package. All packages should be updated eventually, even if that means something is removed because it's no longer needed or something new ends up getting installed.

When you start the process of updating, it will look similar to the following:

A terminal window titled 'jay@ubuntu: ~/terraform' showing the output of the 'sudo apt upgrade' command. The output lists 11 packages to be upgraded, including gir1.2-gtk-4.0, gnome-shell-extension-manager, gstreamer1.0-clutter-3.0, libcephfs2, libclutter-gst-3.0-0, libgtk-4-1, libgtk-4-bin, libgtk-4-common, librados2, librbd1, and xdg-utils. It also indicates that 11 packages will be upgraded, 0 newly installed, 0 to be removed, and 0 not upgraded. The total size of the updates is 14.7 MB, and it will use an additional 154 kB of disk space. The prompt 'Do you want to continue? [Y/n]' is visible at the bottom.

```
jay@ubuntu:~/terraform$ sudo apt upgrade
Reading package lists... Done
Building dependency tree... Done
Reading state information... Done
Calculating upgrade... Done
The following packages will be upgraded:
  gir1.2-gtk-4.0 gnome-shell-extension-manager gstreamer1.0-clutter-3.0 libcephfs2
  libclutter-gst-3.0-0 libgtk-4-1 libgtk-4-bin libgtk-4-common librados2 librbd1
  xdg-utils
11 upgraded, 0 newly installed, 0 to remove and 0 not upgraded.
Need to get 14.7 MB of archives.
After this operation, 154 kB of additional disk space will be used.
Do you want to continue? [Y/n]
```

Figure 21.2: Updating packages on an Ubuntu server

Before the update process actually starts, you'll be given an overview of what it wants to do. In my case, it wants to upgrade 11 packages. If you were to enter `Y` and then press *Enter*, the update process would begin. At this point, I'll need to leave the terminal window open; it's actually dangerous to close it in the middle of the update process. Closing the terminal window in the middle of a package management task may result in corrupted or partially installed packages.

Assuming that this process finishes successfully, we can run the `apt dist-upgrade` command to update the rest – specifically, the packages that were held back because they would've installed new packages or removed existing ones. There weren't any in my case, but in such a situation you may see text indicating that some upgrades were held back, which is normal with `apt upgrade`. At that point, you'll run `sudo apt dist-upgrade` to install any remaining updates that didn't get installed with the first command.

In regard to updating the kernel, this process deserves some additional discussion. Some distributions are very risky when it comes to updating the kernel. Arch Linux is an example of this, where only one kernel is installed at any one time. Therefore, when that kernel gets updated, you really need to reboot the machine so that it can use it properly (sometimes, various system components may have difficulty in a case where you have a pending reboot after installing a new kernel).

Ubuntu, on the other hand, handles kernel upgrades very efficiently. When you update a kernel in Ubuntu, it doesn't replace the kernel your server is currently running on. Instead, it installs the updated kernel alongside your existing one.

In fact, these kernels will continue to be stacked and none of them will be removed as new ones are installed. When new versions of the Ubuntu kernel are installed, the GNU GRUB boot loader will be updated automatically to boot the new kernel the next time you perform a reboot.

Until you do, you can continue to run on your current kernel for as long as you need to, and you shouldn't notice any difference. The only real difference is the fact that you're not taking advantage of the additional security patches of the new kernel until you reboot, which you can do during your next maintenance window. The reason this method of updating is great is that if you run into a problem where the new kernel doesn't boot or has some sort of issue, you'll have a chance to press *Esc* at the beginning of the boot process, where you'll be able to browse a list of all of your installed kernels. Using this list, you can choose between your previous (known, working) kernels and continue to use your server as you were before you updated the kernel. This is a valuable safety net!

After you update the packages on your server, you may want to restart services in order to take advantage of the new security updates. In the case of kernels, you would need to reboot your entire server in order to take advantage of kernel updates, but other updates don't require a reboot. Instead, if you restart the associated service, you'll generally be fine (if the update itself didn't already trigger a restart of a service). For example, if your DNS service (`bind9`) was updated, you would only need to execute the following to restart the service:

```
sudo systemctl restart bind9
```

In addition to keeping packages up to date, it's also important that you understand how to roll back an updated package in a situation where something went wrong. You can recover from such a situation by simply reinstalling an older version of a package manually. Previously downloaded packages are stored in the following directory:

```
/var/cache/apt/archives
```

There, you should find the actual packages that were downloaded as a part of your update process. In a case where you need to restore an updated package to a previously installed version, you can manually install a package with the `dpkg` command. Generally, the syntax will be similar to the following:

```
sudo dpkg -i /path/to/package.deb
```

To be more precise, you would use a command such as the following to reinstall a previously downloaded package, using an older Linux kernel as an example:

```
sudo dpkg -i /var/cache/apt/archives/linux-image-5.15.0-30-  
generic_5.15.0-30.31_amd64.deb
```

However, with the `dpkg` command, dependencies aren't handled automatically, so if you are missing a package that your target package requires as a dependency, the package will still be installed, but you'll have unresolved dependencies you'll need to fix. You can try to resolve this situation with `apt`:

```
sudo apt -f install
```

The `apt -f install` command will attempt to fix your installed packages, looking for packages that are missing (but are required by an installed package), and will offer to install the missing dependencies for you. In a case where it cannot find a missing dependency, it will offer to remove the package that requires the missing packages if the situation cannot be worked out any other way.

Well, there you have it. At this point, you should be well on your way to not only installing packages but keeping them updated as well. There's also a feature in Ubuntu that you can utilize to take advantage of the concept of **live patching**, which you can use to patch your server's kernel automatically. That's what we'll cover in the next section.

Automatically installing patches with the Canonical Livepatch service

In the previous section, I mentioned that if your updates include an update to the kernel, you'll need to reboot your server for the new kernel to take effect. While this is generally true, Canonical offers a **Livepatch** service for Ubuntu, which allows it to receive updates and have them applied without rebooting. This is a game changer, as it takes care of keeping your running kernel patched without you having to do anything, not even reboot. This is a massive benefit to security, as it gives you the benefits of the latest security patches without the inconvenience of scheduling a restart of your servers right away.

However, the service is not free or included with Ubuntu by default. Even so, you can install the Livepatch service on three of your servers without paying, so it's still something you may want to consider. You're even able to utilize this service on the desktop version of Ubuntu if you'd like. Since you can use this service for free on three servers, I see no reason why you shouldn't benefit from this on your most critical resources.

Even though you generally won't need to reboot your server in order to take advantage of patches with the Livepatch service, there may be some exceptions depending on the nature of the vulnerability. There have been exploits in the past that required complex changes, and even servers subscribed to this service still needed to reboot. This is the exception rather than the rule, though. Most of the time, a reboot is simply not something you'll need to worry about if you're utilizing Livepatch. More often than not, your server will have all patches applied and inserted right into the running kernel, which is an amazing thing.

One important thing to note is that this doesn't stop you from needing to install updates via apt. Live patches are inserted right into the kernel, but they're not permanent. You'll still want to install all of your package updates on a regular basis through the regular means. At the very least, live patches will make it so that you won't be in such a hurry to reboot. If an exploit is revealed on Monday but you aren't able to reboot your server until Sunday, it's no big deal.

Since the Livepatch service requires a subscription, you'll need to create an account in order to get started using it. You can get started with this process at <https://auth.livepatch.canonical.com/>.

The process will involve having you create an Ubuntu One account (<https://login.ubuntu.com/>), which is Canonical's centralized login system. You'll enter your email address, choose a password, and then at the end of the process, you'll be given a token to use with your Livepatch service, which will be a string of random characters.

Now that you have a token, you can decide on the three servers that are most important to you. On each of those servers, you can run the following commands to get started:

```
sudo snap install canonical-livepatch
sudo canonical-livepatch enable <token>
```

Believe it or not, that's all there is to it. With how amazing the Livepatch service is, you'd think it would be a complicated process to set up. The most time-consuming part is registering for a new account, as it only takes two commands to set this service up on a server. You can check the status of Livepatch with the following command:

```
sudo canonical-livepatch status
```

Depending on the budget of your organization, you may decide that this service is worth paying for, which will allow you to benefit from having it on more than three servers. It's definitely worth considering. You'll need to contact Canonical to inquire about additional support, should you decide to explore that option.

At this point, we should switch gears and discuss some things we can do to better secure OpenSSH. I've mentioned a few times throughout this chapter that OpenSSH is a common target for outside threat actors, so in the next section, it's time to take a closer look at this.

Securing OpenSSH

OpenSSH is a very useful utility; it allows us to configure our servers from a remote location as if we were sitting in front of the console. In the case of cloud resources, it's typically the only way to access our servers. Considering the nature of OpenSSH itself (remote administration), it's a very tempting target for miscreants who are looking to cause trouble. If we simply leave OpenSSH unsecured, this useful utility may be our worst nightmare.

Thankfully, configuring OpenSSH itself is very easy. However, the large number of configuration options may be intimidating to someone who doesn't have much experience tuning it. While it's a good idea to peruse the documentation for OpenSSH, in this section, we'll take a look at the common configuration options you'll want to focus your attention on first.

The configuration file for OpenSSH itself is located at `/etc/ssh/sshd_config`, and we touched on it in *Chapter 10, Connecting to Networks*. This is the file we're going to focus on in this section, as the configuration options I'm going to give you are to be placed in that file.

With each of the tweaks in this section, make sure you first look through the file in order to see whether the setting is already there and change it accordingly. If the setting is not present in the file, add it. After you make your changes, it's important to restart the OpenSSH daemon:

```
sudo systemctl restart ssh
```

Go ahead and open this file in your editor, and we'll go through some tweaks.

One really easy tweak is to change the port number that OpenSSH listens on, which defaults to port 22. Since this is the first port that hackers will attempt, it makes sense to change it, and it's a very easy change to make. However, I don't want you to think that just because you change the port for OpenSSH, it's magically hidden and cannot be detected. A persistent threat actor will still be able to find the port by running a port scan against your server. However, with the change being so easy to tweak, why not do it? To change it, simply look for the port number in the `/etc/ssh/sshd_config` file and change it from its default of 22:

```
Port 65332
```

The only downsides I can think of in regards to changing the SSH port are that you'll have to remember to specify the port number when using SSH, and you'll have to communicate the change to anyone that uses the server. To specify the port, we use the `-p` option with the `ssh` command:

```
ssh -p 65332 myhost
```

If you're using `scp`, you'll need to use an uppercase `P` instead:

```
scp -P 65332 myfile myserver:/path/to/dir
```

Even though changing the port number won't make your server bulletproof, we shouldn't underestimate the value of doing so. In a hypothetical example where an attacker is scanning servers on the internet for an open port 22, they'll probably skip your server and move on to the next. Only determined attackers that specifically want to break into your server will scan other ports looking for it. This also keeps your log file clean; you'll see intrusion attempts only from miscreants doing aggressive port scans, rather than random bots looking for open ports.

If your server is internet-facing, this will result in far fewer entries in the logs! OpenSSH logs connection attempts in the authorization log, located at `/var/log/auth.log`. Feel free to check out that log file to see what typical logging looks like.

Another change that's worth mentioning is which protocol OpenSSH listens for. Most versions of OpenSSH available in repositories today default to Protocol 2. This is what you want. Protocol 2 is much more secure than Protocol 1. You should never allow Protocol 1 in production under any circumstances. Chances are you're probably already using the default of Protocol 2 on your server, unless you changed it for some reason. I mention it here just in case you have older servers still in production that are defaulting to the older protocol. Nowadays, OpenSSH is always on Protocol 2 in any modern release of a Linux distribution. If you do have an older server that's still using Protocol 1, you can adjust that by finding the following line in the `/etc/ssh/sshd_config` file:

```
Protocol 1
```

Switching OpenSSH to use Protocol 2 is as simple as changing the 1 on that line to 2, and then restarting the OpenSSH server:

```
sudo systemctl restart ssh
```

Next, I'll give you two tweaks for the price of one. There are two settings that deal with which users and groups are allowed to log in via SSH: `AllowUsers` and `AllowGroups`, respectively. By default, every user you create is allowed to log in to your server via SSH. With regards to root, that's actually not allowed by default (more on that later). But each user you create is allowed in. However, only users that must have access should be allowed in. There are two ways to accomplish this.

One option is to use `AllowUsers`. With the `AllowUsers` option, you can specifically set which users can log in to your server. With `AllowUsers` present (which is not found in the `config` file by default), your server will not allow anyone to use SSH that you don't specifically call out with that option. You can separate each user with a space:

```
AllowUsers larry moe curly
```

Personally, I find `AllowGroups` easier to manage. It works pretty much the same as `AllowUsers`, but with groups. If present, it will restrict OpenSSH connections to users who are a member of this group. To use it, you'll first create the group in question (you can name it whatever makes sense to you):

```
sudo groupadd sshusers
```

Then, you'll make one or more users a member of that group:

```
sudo usermod -aG sshusers myuser
```

Once you have added the group and made a user or two a member of that group, add the following to your `/etc/ssh/sshd_config` file, replacing the sample groups with yours:

```
AllowGroups admins sshusers gremlins
```

It's fine to use only one group. Just make sure you add yourself to the group before you log out; otherwise, you'll lock yourself out. I recommend you use only one or the other between `AllowUsers` and `AllowGroups`. I think that it's much easier to use `AllowGroups`, since you'll never need to touch the `sshd_config` file again; you'll simply add or remove user accounts to and from the group to control access. Just so you're aware, `AllowUsers` overrides `AllowGroups`.

Another important option is `PermitRootLogin`, which controls whether or not the root user account is able to make SSH connections. This should always be set to `no`. By default, this is usually set to `prohibit-password`, which means key authentication is allowed for root while passwords for root aren't accepted. I don't see any reason for this either. In my opinion, you should turn this off. Having root being able to log in to your server over a network connection is never a good idea. This is always the first user account attackers will try to use:

```
PermitRootLogin no
```

There is one exception to the *no-root* rule with SSH. Some providers of cloud servers, such as Linode, may have you log in as root by default. This isn't really typical, but some providers are set up that way. In such a case, I recommend creating a regular user with `sudo` access, and then disallowing root login.

My next suggestion is by no means easy to set up, but it's worth it. By default, OpenSSH allows users to authenticate via passwords. This is one of the first things I disable on all my servers. Allowing users to enter passwords to establish a connection means that attackers will also be able to brute-force your server. If passwords aren't allowed, then they can't do that. What's tricky is that before you can disable password authentication for SSH, you'll first need to configure and test an alternate means of authenticating, which will usually be public key authentication. This is something we've gone over, in *Chapter 10, Connecting to Networks*. Basically, you can generate an SSH key pair on your local workstation, and then add that key to the `authorized_keys` file on the server, which will allow you in without a password. Again, refer to *Chapter 10, Connecting to Networks*, if you haven't played around with this yet.

If you disable password authentication for OpenSSH, then public key authentication will be the only way in. If someone tries to connect to your server and they don't have the appropriate key, the server will deny their access immediately. If password authentication is enabled and you have a key relationship, then the server will ask the user for their password if their key isn't installed. In my view, after you set up access via public key cryptography, you should disable password authentication (just make sure you test it first):

```
PasswordAuthentication no
```

There you are – those are my most recommended tweaks for securing OpenSSH. There's certainly more where that came from, but those are the settings you'll benefit from the most. In the next section, we'll add an additional layer, in the form of Fail2ban. With Fail2ban protecting OpenSSH and coupled with the tweaks I mentioned in this section, attackers will have a tough time trying to break into your server. For your convenience, here are all the OpenSSH configuration options I've covered in this section:

```
Port 65332
Protocol 2
AllowUsers larry moe curly
AllowGroups admins sshusers gremlins
PermitRootLogin no
PasswordAuthentication no
```

With OpenSSH better secured, we should be a bit more confident now when it comes to the security of our server. However, each tweak or improvement we make to improve security only helps us so much. The more protections we implement, the better. In the next section, we'll explore Fail2ban, which can greatly increase the security of our server.

Installing and configuring Fail2ban

Fail2ban, how I love thee! Fail2ban is one of those tools that once I learned how valuable it is, I wondered how I ever lived so long without it. Fail2ban is able to keep an eye on your log files, looking for authentication failures. You can set the number of failures that are allowed from any given IP address, and if there are more than the allowed number of failures, Fail2ban will block that individual's IP address. It's highly configurable and can enhance the security of your server.

Installing and configuring Fail2ban is relatively straightforward. First, install its package:

```
sudo apt install fail2ban
```

After installation, the `fail2ban` daemon will start up and be configured to automatically start at boot time. Configuring `fail2ban` is simply a matter of creating a configuration file. But this is one of the more interesting aspects of Fail2ban: you shouldn't use its default config file. The default file is `/etc/fail2ban/jail.conf`. The problem with this file is that it can be overwritten when you install security updates, if those security updates ever include Fail2ban itself. To remedy this, Fail2ban also reads the `/etc/fail2ban/jail.local` file, if it exists. It will never replace that file, and the presence of a `jail.local` file will supersede the `jail.conf` file. The simplest way to get started is to make a copy of `jail.conf` and save it as `jail.local`:

```
sudo cp /etc/fail2ban/jail.conf /etc/fail2ban/jail.local
```

Next, I'll go over some of the very important settings you should configure, so open up the `/etc/fail2ban/jail.local` file you just copied in a text editor. The first configuration item to change is located on or around line 92 and is commented out:

```
#ignoreip = 127.0.0.1/8 ::1
```

First of all, uncomment it. Then, you should add additional networks that you don't want to be blocked by Fail2ban. Basically, this will help prevent you from getting locked out in a situation where you accidentally trigger Fail2ban. Fail2ban is relentless; it will block any service that meets its block criteria, and it won't think twice about it. This includes blocking you. To rectify this, add your company's network here, as well as any other IP address you never want to be blocked. Make sure to leave the localhost IP intact:

```
Ignoreip = 127.0.0.1/8 ::1 192.168.1.0/24 192.168.1.245/24
```

In that example, I added the `192.168.1.0/24` network, as well as a single IP address of `192.168.1.245/24`. Add your networks to this line to *ensure you don't lock yourself out*.

Next, line 101 includes the `bantime` option. This option pertains to how many seconds a host is banned when Fail2ban blocks it. This option defaults to `10m`, or 10 minutes:

```
bantime = 10m
```

Change this number to whatever you find reasonable, or just leave it as its default, which will also be fine. If a host gets banned, it will be banned for this specific number of minutes, and then it will eventually be allowed again.

Continuing, we have the `maxretry` setting:

```
maxretry = 5
```

This is specifically the number of failures that need to occur before Fail2ban takes action. If a service it's watching reaches the number set here, game over! The IP will be blocked for the number of minutes included in the `bantime` option.

You can change this if you want to, if you don't find 5 failures to be reasonable. The highest I would set it to is 7, for those users on your network who insist they're typing the correct password and they type the same (wrong) thing over and over. Hopefully, they'll realize their error before their seventh attempt and won't need to call the helpdesk.

Skipping ahead all the way down to line 272 or thereabouts, we have the `Jails` section. From here, the config file will list several jails you can configure, which is basically another word for something Fail2ban will pay attention to. The first is `[sshd]`, which configures its protection of the OpenSSH daemon. Look for this option underneath `[sshd]`:

```
port = ssh
```

`port` being equal to `ssh` basically means that it's defaulting to port 22. If you've changed your SSH port, change this to reflect whatever that port is. There are two such occurrences, one under `[sshd]` and another underneath `[sshd-ddos]`:

```
port = 65332
```

Before we go too much further, I want to underscore the fact that we should test whether Fail2ban is working after each configuration change we make. To do this, restart Fail2ban and then check its status:

```
sudo systemctl restart fail2ban
sudo systemctl status -l fail2ban
```

The status should always be `active (running)`. If it's anything else (such as `failed`), that means that Fail2ban doesn't like something in your configuration. Usually, that means that Fail2ban's status will reflect that it exited. So, as we go, make sure to restart Fail2ban after each change and make sure it's not complaining about something. The `status` command will show lines from Fail2ban's log file for your convenience.

Another useful command to run after restarting Fail2ban is the following:

```
sudo fail2ban-client status
```

The output from that command will show all the jails that you have enabled. If you enable a new jail in the config file, you should see it listed within the output of that command.

So, how do you enable a jail? By default, all jails are disabled, except for the one for OpenSSH. To enable a jail, place the following within its config block in the `/etc/fail2ban/jail.local` file:

```
enabled = true
```

If you want to enable the `apache-auth` jail, find its section, and place `enabled = true` right underneath its stanza. For example, `apache-auth` will look like the following after you add the `enabled` line:

```
[apache-auth]
enabled = true
port     = http,https
logpath  = %(apache_error_log)
```

In that example, the `enabled = true` portion wasn't present in the default file. I added it. Now that I've enabled a new jail, we should restart `fail2ban`:

```
sudo systemctl restart fail2ban
```

Next, check its status to make sure it didn't explode on startup:

```
sudo systemctl status -l fail2ban
```

Assuming all went well, we should see the new jail listed in the output of the following command:

```
sudo fail2ban-client status
```

On my test server, the output became the following once I enabled `apache-auth`:

```
Status
|- Number of jail: 2
  '- Jail list:   apache-auth, sshd
```

If you enable a jail for a service you don't have installed, Fail2ban may fail to start up. In my example, I actually did have `apache2` installed on that server before I enabled its jail. If I hadn't, Fail2ban would likely have exited, complaining that it wasn't able to find log files for Apache. This is the reason why I recommend that you test Fail2ban after enabling any jail. If Fail2ban decides it doesn't like something, or something it's looking for isn't present, it may stop. Then, it won't be protecting you at all, which is not good.

The basic order of operations for Fail2ban is to peruse the jail config file, looking for any jails you may benefit from. If you have a daemon running on your server, there's a chance that there's a jail for that. If there is, enable it and see whether Fail2ban breaks. If not, you're in good shape. If it does fail to restart properly, inspect the status output and check what it's complaining about.

One thing you may want to do is add the `enabled = true` line to `[sshd]` and `[sshd-ddos]`. Sure, the `[sshd]` jail is already enabled by default, but since it wasn't specifically called out in the config file, I don't trust it. So you might as well add an `enabled` line to be safe. There are several jails you may benefit from. If you are using SSL with Apache, enable `[apache-modsecurity]`. Also, consider enabling `[apache-shellshock]` while you're at it to potentially protect Apache from the Shellshock vulnerability. If you're running your own mail server and have Roundcube running, enable `[roundcube-auth]` and `[postfix]`. There are a lot of default jails at your disposal!

Like all security applications, Fail2ban isn't going to automatically make your server impervious to all attacks, but it is a helpful additional layer you can add to your security regimen. When it comes to the jails for OpenSSH, Fail2ban is worth its weight in gold, and that's really the least you should enable. Go ahead and give Fail2ban a go on your servers—just make sure you also add your own network to the `Ignoreip` list that was covered earlier, in case you accidentally type your own SSH password incorrectly too many times and potentially lock yourself out. Fail2ban doesn't discriminate; it'll block anyone. Once you get it fully configured, I think you'll agree that Fail2ban is a worthy ally for your servers.

Earlier, I mentioned that each service that runs on your computer listening for connections is a potential target. While it's impossible to go over every service you could possibly run on your server and how to secure it, we will want to consider securing our database server (if we have one) since organizations typically store valuable data there. We'll learn some methods we can utilize to better secure MariaDB next.

MariaDB best practices for secure database servers

MariaDB, as well as MySQL, is a very useful resource to have at your disposal. However, it can also be used against you if configured improperly. Thankfully, it's not too hard to secure, but there are several points of consideration to make regarding your database server when developing your security design.

The first point is probably obvious to most of you, and I have mentioned it before, but I'll mention it just in case. Your database server should not be reachable from the internet. I do understand that there are some edge cases when developing a network, and certain applications may require access to a MySQL database over the internet. However, if your database server is accessible over the internet, miscreants will try their best to attack it and gain entry. If there's any vulnerability in your version of MariaDB or MySQL, they'll most likely be able to hack into it.

In most organizations, a great way to implement a database server is to make it accessible by only internal servers. This means that while your web server would obviously be accessible from the internet, its backend database should exist on a different server on your internal network and accept communications only from the web server. If your database server is a VPS or cloud instance, it should especially be configured to only accept communications from your web server, as VPS machines are accessible via the internet by default. Therefore, it's still possible for your database server to be breached if your web server is also breached, but it would be less likely to be compromised if it resides on a separate and restricted server.

Some VPS providers, such as DigitalOcean and Linode, feature local networking, which you can leverage for your database server instead of allowing it to be accessible over the internet. If your VPS provider features local networking, you should definitely utilize it and deny traffic from outside the local network.

With regards to limiting which servers are able to access a database server, there are a few tweaks we can use to accomplish this. First, we can leverage the `/etc/hosts.allow` and `/etc/hosts.deny` files. With the `/etc/hosts.deny` file, we can stop traffic from certain networks or from specific services. With `/etc/hosts.allow`, we allow the traffic. This works because IP addresses included in `/etc/hosts.allow` override `/etc/hosts.deny`. So basically, if you deny everything in `/etc/hosts.deny` and allow a resource or two in `/etc/hosts.allow`, you're saying, deny everything, except resources I explicitly allow from the `/etc/hosts.deny` file.

To make this change, we'll want to edit the `/etc/hosts.allow` file first. By default, this file has no configuration other than some helpful comments. Within the file, we can include a list of resources we'd like to be able to access our server, no matter what. Make sure that you include your web server here, and also make sure that you immediately add the IP address you'll be using to SSH into the machine; otherwise, you'll lock yourself out once we edit the `/etc/hosts.deny` file.

Here are some example `hosts.allow` entries, with a description of what each example rule does.

The first example rule allows a machine with an IP address of `192.168.1.50` to access the server:

```
ALL: 192.168.1.50
```

This rule allows any machine within the `192.168.1.0/24` network to access the server:

```
ALL: 192.168.1.0/255.255.255.0
```

In this rule, we have an incomplete IP address. This acts as a wildcard, which means that any IP address beginning with `192.168.1` is allowed:

```
ALL: 192.168.1.
```

This rule allows everything. You definitely don't want to do this:

```
ALL: ALL
```

We can also allow specific daemons. Here, I'm allowing OpenSSH traffic originating from any IP address beginning with `192.168.1`:

```
ssh: 192.168.1.
```

On your end, if you wish to utilize this security approach, add the resources on the database server you'll be comfortable accepting communications from. Make sure you at least add the IP address of another server with access to OpenSSH, so you'll have a way to manage the machine. You can also add all your internal IP addresses with a rule similar to the previous examples. Once you have this set up, we can edit the `/etc/hosts.deny` file.

The `/etc/hosts.deny` file utilizes the same syntax as `/etc/hosts.allow`. To finish this little exercise, we can block any traffic not included in the `/etc/hosts.allow` file with the following rule:

```
ALL: ALL
```

The `/etc/hosts.allow` and `/etc/hosts.deny` files don't represent a complete layer of security but are a great first step in securing a database server, especially one that might contain sensitive user or financial information. They're by no means specific to MariaDB either, but I mention them here because databases very often contain data that, if leaked, could potentially wreak havoc on your organization and even put someone out of business. A database server should only ever be accessible by the application that needs to utilize it.

Another point of consideration is user security. We walked through creating database users in *Chapter 13, Managing Databases*. In that chapter, we walked through the MySQL commands for creating a user as well as `GRANT`, performing both in one single command. This is the example I used:

```
GRANT SELECT ON mysampledb.* TO 'appuser'@'localhost' IDENTIFIED BY 'password';
```

What's important here is that we're allowing access to the `mysampledb` database by a user named `appuser`. If you look closer at the command, we're also specifying that this connection is allowed only if it's coming in from `localhost`. If we tried to access this database remotely, it wouldn't be allowed.

This is a great default. But you'll also, at some point, need to access the database from a different server. Perhaps your web server and database server are separate machines, which is a common enterprise. You could do this:

```
GRANT SELECT ON mysampledб.* TO 'appuser'@'%' IDENTIFIED BY 'password';
```

However, in my opinion, this is a very bad practice. The % character in a MySQL GRANT command is a wildcard, similar to * with other commands. Here, we're basically telling our MariaDB or MySQL instance to accept connections from this user, from any network. There is almost never a good reason to do this. I've heard some administrators use the argument that they don't allow external traffic from their company firewall, so allowing MySQL traffic from any machine shouldn't be a problem. However, that logic breaks down when you consider that if an attacker does gain access to any machine in your network, they can immediately target your database server. If an internal employee gets angry at management and wants to destroy the database, they'll be able to access it from their workstation. If an employee's workstation becomes affected by malware that targets database servers, it may find your database server and try to brute-force it. I could go on and on with examples of why allowing access to your database server from any machine is a bad idea. Just don't do it!

If we want to give access to a specific IP address, we can do so with the following instead:

```
GRANT SELECT ON mysampledб.* TO 'appuser'@'192.168.1.50' IDENTIFIED BY 'password';
```

With the previous example, only a server or workstation with an IP address of 192.168.1.50 is allowed to use the appuser account to obtain access to the database. That's much better. You can, of course, allow an entire subnet as well:

```
GRANT SELECT ON mysampledб.* TO 'appuser'@'192.168.1.%' IDENTIFIED BY 'password';
```

Here, any IP address beginning with 192.168.1 is allowed. Honestly, I really don't like allowing an entire subnet. But depending on your network design, you may have a dozen or so machines that need access. Hopefully, the subnet you allow is not the same subnet your users' workstations use!

Finally, another point of consideration is security patches for your database server software. I know I talk about updates quite a bit, but as I've mentioned, these updates exist for a reason. Developers don't release patches for enterprise software simply because they're bored; these updates often patch real problems that real people are taking advantage of right now as you read this. Install updates regularly. I understand that updates on server applications can scare some people, as an update always comes with the risk that it may disrupt business. But as an administrator, it's up to you to create a rollout plan for security patches, and ensure they're installed in a timely fashion. Sure, it's tough and often has to be done after hours. But the last thing I want to do is read about yet another company where the contents of their database server were leaked and posted freely online. A good security design includes regular patching.

Now that our database server is more secure, there's another topic worth diving into, and that is the subject of implementing a firewall. There are several different firewall solutions out there, but UFW is a great choice. It's easy to set up, and quite effective. In the next section, I'll go over how to implement it.

Setting up a firewall

Firewalls are a very important aspect to include in your network and security design. Firewalls are extremely easy to implement, but sometimes hard to implement well. The problem with firewalls is that they can sometimes offer a false sense of security to those who aren't familiar with the best ways to manage them. Sure, they're good to have, but simply having a firewall isn't enough by itself.

The false sense of security comes when someone thinks that they're protected just because a firewall is installed and enabled, but they're also often opening traffic from any network to internal ports. Take into consideration the firewall that was introduced with Windows XP and enabled by default with Windows XP Service Pack 2. Yes, it was a good step but users simply clicked the **allow** button whenever something wanted access, which defeats the entire purpose of having a firewall. Windows implements this better nowadays, but the false sense of security it created remains. Firewalls are not a "set it and forget it" solution!

Firewalls work by allowing or disallowing access to a network port from other networks. Most good firewalls block outside traffic by default. When a user or administrator enables a service, they open a port for it. Then, that service is allowed in. This is great in theory, but where it breaks down is that administrators will often allow access from everywhere when they open a port. If an administrator does this, they may as well not have a firewall at all. If you need access to a server via OpenSSH, you may open up port 22 (or whatever port OpenSSH is listening on) to allow it through the firewall. But if you simply allow the port, it's open for everyone else as well.

When configured properly, a firewall will enable access to a port only from specific places. For example, rather than allowing port 22 for OpenSSH to your entire network, why not just allow traffic to port 22 from specific IP addresses or subnets? Now we're getting somewhere! In my opinion, allowing all traffic through a port is usually a bad idea, though some services actually do need this (such as web traffic to your web server). If you can help it, only allow traffic from specific networks when you open a port. This is where the use case for a firewall really shines.

In Ubuntu Server, **Uncomplicated Firewall (UFW)** is a really useful tool for configuring your firewall. As the name suggests, it makes firewall management a breeze. To get started, install the `ufw` package:

```
sudo apt install ufw
```

By default, the UFW is inactive. This is a good thing, because we wouldn't want to enable a firewall until after we've configured it. The `ufw` package features its own command for checking its status:

```
sudo ufw status
```

Unless you've already configured your firewall, the status will come back as inactive.

With the `ufw` package installed, the first thing we'll want to do is enable traffic via SSH, so we won't get locked out when we do enable the firewall:

```
sudo ufw allow from 192.168.1.156 to any port 22
```

You can probably see from that example how easy UFW's syntax is. With that example, we're allowing the 192.168.1.156 IP address access to port 22 via TCP as well as UDP. In your case, you would change the IP address accordingly, as well as the port number if you're not using the OpenSSH default port. The any option refers to any protocol (TCP or UDP).

You can also allow traffic by subnet:

```
sudo ufw allow from 192.168.1.0/24 to any port 22
```

Although I don't recommend this, you can allow all traffic from a specific IP to access anything on your server. Use this with care, if you have to use it at all:

```
sudo ufw allow from 192.168.1.50
```

Now that we've configured our firewall to allow access via OpenSSH, you should also allow any other ports or IP addresses that are required for your server to operate efficiently. If your server is a web server, for example, you'll want to allow traffic from ports 80 and 443. This is one of those few exceptions where you'll want to allow traffic from any network, assuming your web server serves an external page on the internet:

```
sudo ufw allow 80
sudo ufw allow 443
```

There are various other use patterns for the ufw command; refer to the main page (<http://manpages.ubuntu.com/manpages/focal/man8/ufw.8.html>) for more. In a nutshell, these examples should enable you to allow traffic through specific ports, as well as via specific networks and IP addresses. Once you've finished configuring the firewall, we can enable it:

```
sudo ufw enable
Firewall is active and enabled on system startup
```

Just as the output suggests, our firewall is active and will start up automatically whenever we reboot the server.

The UFW package is basically an easy-to-use frontend to the iptables firewall, and it acts as the default firewall for Ubuntu. The commands we've executed so far in this section trigger the iptables command, which is a command that administrators can use to set up a firewall manually. A full walk-through of iptables is outside the scope of this chapter, and it's essentially unnecessary, since Ubuntu features UFW as its preferred firewall administration tool and it's the tool you should use while administering a firewall on your Ubuntu server.

With a well-planned firewall implementation, you can better secure your Ubuntu Server installation from outside threats. Preferably, each port you open should only be accessible from specific machines, with the exception being servers that are meant to serve data or resources to external networks. Like all security solutions, a firewall won't make your server invincible, but it does represent an additional layer that attackers would have to bypass in order to do harm.

If your company stores sensitive information, it's important to ensure the storage underneath that data is encrypted. Next, we're going to look at **Linux Unified Key Setup (LUKS)**, which will help us encrypt and decrypt disks.

Encrypting and decrypting disks with LUKS

An important aspect of security that many people don't even think about is encryption. As I'm sure you know, backups are essential for business continuity. If a server breaks down, or a resource stops functioning, backups will be your saving grace. But what happens if your backup medium gets stolen or somehow falls into the wrong hands? If your backup is not encrypted, then anyone will be able to view its contents. Some data isn't sensitive, so encryption isn't always required. But anything that contains personally identifiable information, company secrets, or anything else that would cause any kind of hardship if leaked should be encrypted. In this section, I'll walk you through setting up **LUKS** encryption on an external backup drive.

Before we get into that though, I want to quickly mention the importance of full-disk encryption for your distribution as well. Although this section is going to go over how to encrypt external disks, it's possible to encrypt the volume for your entire Linux installation as well. In the case of Ubuntu, full-disk encryption is an option during installation, for both the server and workstation flavors. This is especially important when it comes to mobile devices, such as laptops, which are stolen quite frequently. If a laptop is planned to store confidential data that you cannot afford to have leaked out, you should choose the option during installation to encrypt your entire Ubuntu installation. If you don't, anyone that knows how to boot a Live OS disc and mount a hard drive will be able to view your data. I've seen unencrypted company laptops get stolen before, and it's not a wonderful experience.

Anyway, back to the topic of encrypting external volumes. For the purpose of encrypting disks, we'll need to install the `cryptsetup` package:

```
sudo apt install cryptsetup
```

The `cryptsetup` utility allows us to encrypt and unencrypt disks. To continue, you'll need an external disk you can safely format, as encrypting the disk will remove any data stored on it. This can be an external hard disk or a flash drive. Both can be treated the exact same way. In addition, you can use this same process to encrypt a secondary internal hard disk attached to your virtual machine or server. I'm assuming that you don't care about the contents saved on the drive, because the process of setting up encryption will wipe it.

If you're using an external disk, use the `fdisk -l` command as root or the `lsblk` command to view a list of hard disks attached to your computer or server before you insert it. After you insert your external disk or flash drive, run the command again to determine the device designation for your removable media.

In my examples, I used `/dev/sdb`, but you should use whatever designation your device was given. This is important, because you don't want to wipe out your root partition or an existing data partition!

First, we'll need to use `cryptsetup` to format our disk:

```
sudo cryptsetup luksFormat /dev/sdb
```

You'll receive the following warning:

```
WARNING!  
=====  
This will overwrite data on /dev/sdb irrevocably.  
Are you sure? (Type uppercase yes):
```

Type YES and press *Enter* to continue. Next, you'll be asked for the passphrase. This passphrase will be required in order to unlock the drive. Make sure you use a good, randomly generated password and that you store it somewhere safe. If you lose it, you will not be able to unlock the drive. You'll be asked to confirm the passphrase.

Once the command completes, we can format our encrypted disk. At this point, it has no filesystem, so we'll need to create one. First, open the disk with the following command:

```
sudo cryptsetup luksOpen /dev/sdb backup_drive
```

The `backup_drive` name can be anything you want; it's just an arbitrary name you can refer to the disk as. At this point, the disk will be attached to `/dev/mapper/disk_name`, where `disk_name` is whatever you called your disk in the previous command (in my case, `backup_drive`). Next, we can format the disk. The following command will create an ext4 filesystem on the encrypted disk:

```
sudo mkfs.ext4 -L "backup_drive" /dev/mapper/backup_drive
```

The `-L` option allows us to add a label to the drive, so feel free to change that label to whatever you prefer to name the drive.

With the formatting out of the way, we can now mount the disk:

```
sudo mount /dev/mapper/backup_drive /media/backup_drive
```

The `mount` command will mount the encrypted disk located at `/dev/mapper/backup_drive` and attach it to a mount point, such as `/media/backup_drive` in my example. The target mount directory must already exist. With the disk mounted, you can now save data onto the device as you would any other volume. When finished, you can unmount the device with the following commands:

```
sudo umount /media/backup_drive  
sudo cryptsetup luksClose /dev/mapper/backup_drive
```

First, we unmount the volume just like we normally would. Then, we tell `cryptsetup` to close the volume. To mount it again, we would issue the following commands:

```
sudo cryptsetup luksOpen /dev/sdb backup_drive  
sudo mount /dev/mapper/backup_drive /media/backup_drive
```

The first of those commands should prompt you for your passphrase. If successful, you can use the second of those commands to mount the volume.

If we wish to change the passphrase, we can use the following command. The disk must not be mounted or open in order for this to work:

```
sudo cryptsetup luksChangeKey /dev/sdb -S 0
```

The command will ask you for the current passphrase, and then the new one twice.

Keep in mind that you should absolutely be careful typing in the new passphrase, so that you don't lock yourself out of the drive.

That's basically all there is to it. With the `cryptsetup` utility, you can set up your own LUKS-encrypted volumes for storing your most sensitive information. If the disk ever falls into the wrong hands, it won't be as bad a situation as it would have been if the disk had been unencrypted. Breaking a LUKS-encrypted volume would take considerable effort that wouldn't be feasible.

In the next section, we'll explore how we can lock down `sudo`. Since `sudo` is an essential command that gives us the ability to run tasks as other users, we'll want to be sure to lock that down too.

Locking down sudo

We've been using the `sudo` command throughout the book. In fact, we took a deeper look at it in *Chapter 2, Managing Users and Permissions*. Therefore, I won't go into too much detail regarding `sudo` here, but some things bear repeating as `sudo` has a direct impact on security.

First and foremost, access to `sudo` should be locked down as much as possible. A user with full `sudo` access is a threat, plain and simple. All it would take is for someone with full `sudo` access to make a single mistake with the `rm` command to cause you to lose data or render your entire server useless. After all, a user with full `sudo` access can do anything root can do (which is everything).

By default, the user you've created during installation will be made a member of the `sudo` group. Members of this group have full access to the `sudo` command. Therefore, you shouldn't make any users a member of this group unless you absolutely have to. In *Chapter 2, Managing Users and Permissions*, I talked about how to control access to `sudo` with the `visudo` command; refer to that chapter for a refresher if you need it. In a nutshell, you can lock down access to `sudo` to specific commands, rather than allowing your users to do everything. For example, if a user needs access to shut down or reboot a server, you can give them access to perform those tasks (and only those tasks) with the following setting:

```
charlie ALL=(ALL:ALL) /usr/sbin/reboot,/usr/sbin/shutdown
```

For the most part, if a user needs access to `sudo`, just give them access to the specific commands that are required as part of their job. If a user needs access to work with removable media, give them `sudo` access for the `mount` and `umount` commands. If they need to be able to install new software, give them access to the `apt` suite of commands, and so on. The fewer permissions you give a user, the better. This goes all the way back to the principle of least privilege that we went over near the beginning of this chapter.

Although most of the information in this section is not new to anyone who has already read *Chapter 2, Managing Users and Permissions*, sudo access is one of those things a lot of people don't think about when it comes to security. The sudo command with full access is equivalent to giving someone full access to the entire server. Therefore, it's an important thing to keep in mind when it comes to hardening the security of your network.

Summary

In this chapter, we looked at the ways in which we can harden the security of our server. A single chapter or book can never give you an all-inclusive list of all the security settings you could possibly configure, but the examples we worked through in this chapter are a great starting point. Along the way, we looked at the concepts of lowering your attack surface, as well as the principle of least privilege. We also looked into securing OpenSSH, which is a common service that many attackers will attempt to use in their favor.

We also looked into Fail2ban, which is a handy daemon that can block other nodes when there are a certain number of authentication failures. We also discussed configuring our firewall, using the UFW utility. Since data theft is also unfortunately common, we covered encrypting our backup disks.

In the next chapter, we'll take a look at troubleshooting our server when things go wrong.

Further reading

- Landscape documentation: <https://learnlinux.link/ls-docs>
- Fail2ban manual: <https://learnlinux.link/f2b-man>
- sshd_config file guide: <https://learnlinux.link/sshd-config-doc>
- Ubuntu CVE tracker: <https://learnlinux.link/u-cve>
- Password haystacks (find out how secure your password is):
<https://learnlinux.link/haystack>
- SECURITY NOW (a very informative security podcast): <https://learnlinux.link/sn-podcast>
- ShieldsUP! (a useful tool to see which ports your router has open):
<https://learnlinux.link/grc-su>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaz0>



22

Troubleshooting Ubuntu Servers

So far, we've covered many topics surrounding Ubuntu Server and worked on some really fun projects. We've set up web servers, built automation, and even created infrastructure in the cloud. As the applications and services you've implemented age, your organization may depend on them more and more. But what happens if something your organization relies on suddenly becomes unavailable? What do you do when things don't quite go according to plan?

While it's impossible for us to account for every possible problem that may come up, there are some common places to look for clues when you run into a problem. In this chapter, we'll take a look at some common starting points and techniques that you can utilize when it comes to troubleshooting issues with your servers. Building solid troubleshooting skills is an important focus, and with the concepts explored here, you'll be well on your way.

In this chapter, we will cover:

- Evaluating the scope
- Conducting a root cause analysis
- Viewing system logs
- Tracing network issues
- Troubleshooting resource issues
- Diagnosing defective RAM

The first step with regards to troubleshooting is to analyze the problem and determine how critical a problem it may be. In the next section, we'll explore how to do just that.

Evaluating the scope

When a problem occurs within your servers or network, your systems will exhibit one or more symptoms. Perhaps an application is much slower than normal, maybe users are unable to access the network, or a server suffers from total failure. There are many problems that can come up at any time, and it can be challenging to keep up.

Once you've identified the symptoms of the problem, the next goal is to identify the overall scope. Essentially, this means determining (as best you can) where the problem is most likely to reside, and how many systems and services are affected. Sometimes the root cause is obvious. For example, if none of your computers are receiving an IP address from your DHCP server, then you'll know straight away to start investigating the logs on that particular server concerning its ability (or inability) to do the job designated for it. In other cases, the cause may not be so obvious. Perhaps you have an application that exhibits problems every now and then but isn't something you can reliably reproduce. In that case, it may take some digging before you know just how large the scope of the problem might be. Sometimes, the culprit is the last thing you expect.

Each component on your network works together with other components, or at least that's how it should be. A network of Linux servers, just as with any other network, is a collection of services (daemons) that complement and often depend upon one another. For example, DHCP assigns IP addresses to all of your hosts, but it also assigns their default DNS servers as well. If your DNS server has encountered an issue, then your DHCP server would essentially be assigning a non-working DNS server to your clients. Identifying the problem space means that after you identify the symptoms, you'll also work toward reaching an understanding of how each component within your network contributes to (or is affected by) the problem.

With regards to the scope, we identify how far the problem reaches, as well as how many users or systems are affected by the issue. Perhaps just one user is affected, or an entire subnet. This will help you determine the priority of the issue and decide whether this is something essential that you need to fix now, or something that can wait until later. Often, prioritizing is half the battle; sometimes a user will even be under the impression that their issues are more important than others. Use your best judgment.

When identifying the scope, you'll want to answer the following questions as best as you can:

- What are the symptoms of the issue?
- When did this problem first occur?
- Were there any changes made within the network around that time?
- Has this problem happened before? If so, what was done to fix it last time?
- Which servers or nodes are impacted by this issue?
- How many users are impacted?

If the problem is limited to a single machine, then a few really good places to start poking around is to check who is logged in to the server and which commands have recently been entered. Quite often, I've found the culprit just by checking the Bash history for logged-on users (or users that have recently logged in). With each user account, there should be a `.bash_history` file in their home directory. Within this file is a list of commands that were recently entered. Check this file and see if anyone modified anything recently. I can't tell you how many times this alone has led directly to the answer. And what's even better, sometimes the Bash history leads to the solution. If a problem has occurred before and someone has already fixed it at some point in the past, chances are their efforts were recorded in the Bash history, so you can see what the previous person did to solve the problem just by looking at it.

To view the Bash history, you can either view the contents of the `.bash_history` file in a user's home directory, or you can simply execute the history command as that user.

Additionally, if you check who is currently logged into the server, you may be able to pinpoint if someone is working on an issue already, or perhaps something they're doing caused the issue in the first place. If you enter the `w` command, you can see who is logged in to the server currently. In addition, you'll also see the IP address of the user that's logged in when you run this command. Therefore, if you don't know who corresponds to a user account listed when you run the `w` command, you can check the IP address in your DHCP server to find out who the IP address belongs to, so you can ask that person directly. In a perfect world, other administrators will send out a departmental email when they work on something to make sure everyone is aware. Unfortunately, many don't do this. By checking the logged-in users as well as their Bash history, you're well on your way to determining where the problem originated.

After identifying the problem space and the scope, you can begin narrowing down the issue to help find a cause. Sometimes, the culprit will be obvious. If a website stopped working and you noticed that the Apache configuration on your web server was changed recently, you can attack the problem by investigating the change and who made it.

If the problem is a network issue, such as users not being able to visit websites, the potential problem space is much larger. Your internet gateway may be malfunctioning, your DNS or DHCP server may be down, your internet provider could be having issues, or perhaps your accounting department simply forgot to pay the internet bill. As long as you are able to determine a potential list of targets to focus your troubleshooting on, you're well on your way to finding the issue. As we go through this chapter, I'll talk about some common issues that can come up and how to deal with them.

Understanding the scope of the problem helps us understand just how severe it may be and the number of systems and users impacted, and sometimes, investigating the scope can lead you to the root cause of the problem. If you don't already know the underlying cause, you can conduct a root cause analysis to attempt to find the source of the problem. That's what we'll explore next.

Conducting a root cause analysis

Once you resolve a problem on your server or network, you'll immediately revel in the awesomeness of your troubleshooting skills. It's a wonderful feeling to have fixed an issue, becoming the hero within your technology department. But you're not done yet. The next step is looking toward preventing this problem from happening again. It's important to look at how the problem started as well as steps you can take in order to help stop the problem from occurring again. This is known as a **root cause analysis**. A root cause analysis may be a report you file with your manager or within your knowledge-base system, or it could just be a memo you document for yourself. Either way, it's an important learning opportunity.

A good root cause analysis has several sides to the equation. First, it will demonstrate the events that led to the problem occurring in the first place. Then, it will contain a list of steps that you've completed to correct the problem. If the problem is something that could potentially recur, you would want to include information about how to prevent it from happening again in the future.

The problem with a root cause analysis is that it's rare that you can be 100 percent accurate. Sometimes, the root cause may be obvious. For example, suppose a user named Bob deleted an entire directory that contained files important to your company. If you log into the server and check the logs, you can see that Bob not only logged into the server near the time of the incident, but his Bash history literally shows him running the `rm -rf /work/important-files` command. At this point, the case is closed. You figured out how the problem happened and who did it, and you can restore the files from your most recent backup. But a root cause is usually not that cut and dry.

One example I've personally encountered was a pair of **Virtual Machine (VM)** servers that were "fencing." At a company I once worked for, our Citrix-based VM servers (which were part of a cluster) both went down at the same time, taking every Linux VM down with them. When I attached a monitor to them, I could see them both rebooting over and over. After I got the servers to settle down, I started to investigate deeper. I read in the documentation for Citrix XenServer that you should never install a cluster of anything less than three machines because it can create a situation exactly like the one I experienced. We only had two servers in that cluster, so I concluded that the servers were set up improperly and the company would need a third server if they wanted to cluster them.

The problem though is that this root cause analysis wasn't 100 percent perfect. Were the servers having issues because they needed a third server? The documentation did mention that three servers were a minimum, but there's no way to know for sure that was the reason the problem started. However, not only was I not watching the servers when it happened, but I also wasn't the individual who set them up; that person had already left the company. There was no way I could reach an absolute conclusion, but my root cause analysis was sound in the sense that it was the most likely explanation (that we weren't using best practices). Someone could counter my root cause analysis with "but the servers were running fine that way for several years." True, but nothing is absolute when dealing with technology. Sometimes, you never really know. The only thing you can do is make sure everything is set up properly according to the guidelines set forth by the manufacturer.

A good root cause analysis is as sound in logic as it can be, though not necessarily bulletproof. Correlating system events to symptoms is often a good first step but is not necessarily perfect. After investigating the symptoms, solving the issue, and documenting what you've done to rectify it, sometimes the root cause analysis writes itself. Other times, you'll need to read the documentation and ensure that the configuration of the server or daemon that failed was implemented along with best practices. In a worst-case scenario, you won't really know how the problem happened or how to prevent it, but it should still be documented in case other details come to light later. And without documentation, you'll never gain anything from the situation.

A root cause analysis should include details such as the following:

- A description of the issue
- Which application or piece of hardware encountered a fault

- The date and time the issue was first noticed
- What you found while investigating the issue
- What you've done to resolve the issue
- What events, configurations, or faults caused the issue to happen

A root cause analysis should be used as a learning experience. Depending on what the issue was, it may serve as an example of what not to do, or what to do better. In the case of my VM server fiasco, the moral of the story was to follow best practices from Citrix and use three servers for the cluster instead of two. Other times, the end result may be another technician not following proper directives or making a mistake, which is unfortunate. In the future, if the issue were to happen again, you'll be able to look back and remember exactly what happened last time and what you did to fix it. This is valuable, if only because we're all human and prone to forgetting important details after a time. In an organization, a root cause analysis is valuable to show stakeholders that you're able to not only address a problem but are reasonably able to prevent it from happening again.

Often, log files are a great place to find clues, as quite a bit of information surrounding system and application events are stored there. In the next section, we'll explore log files in more detail.

Viewing system logs

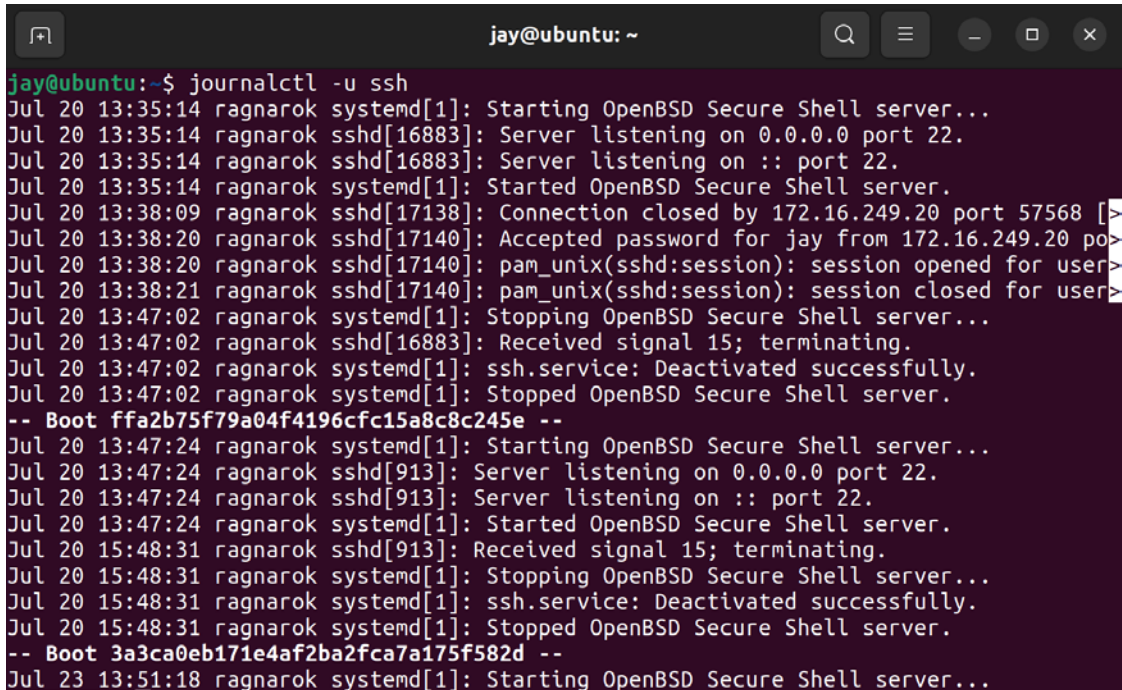
If you're having trouble finding the root cause, or you just want more information regarding a problem that occurred, consider looking through log files. Linux has great logging capabilities, and many of the applications you may be running are writing log files as events happen. If there's an issue, you may be able to find information about it within an application's logs.

There are two primary methods of viewing logs. Historically, for most of Ubuntu's life, you could simply inspect the log files that are stored within the `/var/log` directory. The files contained within that directory are standard files and directories, so you can use commands you've used in the past to view the contents of text files to view the contents of the log files within the `/var/log` directory as well. This method of viewing log files is slowly being aged out; however, the majority of applications still store their log files within that directory, even today.

The newer method of viewing logging information for an application is to use the `journalctl` command. This command is part of `systemd`, and it's dedicated to the purpose of viewing logs. To use the `journalctl` command to check the status of a running service, you provide it the `-u` option, along with the name of a service you'd like to check:

```
journalctl -u ssh
```

With that example, we're attempting to view logging information for the ssh service:



```

jay@ubuntu: ~
jay@ubuntu:~$ journalctl -u ssh
Jul 20 13:35:14 ragnarok systemd[1]: Starting OpenBSD Secure Shell server...
Jul 20 13:35:14 ragnarok sshd[16883]: Server listening on 0.0.0.0 port 22.
Jul 20 13:35:14 ragnarok sshd[16883]: Server listening on :: port 22.
Jul 20 13:35:14 ragnarok systemd[1]: Started OpenBSD Secure Shell server.
Jul 20 13:38:09 ragnarok sshd[17138]: Connection closed by 172.16.249.20 port 57568 [v
Jul 20 13:38:20 ragnarok sshd[17140]: Accepted password for jay from 172.16.249.20 po
Jul 20 13:38:20 ragnarok sshd[17140]: pam_unix(sshd:session): session opened for user
Jul 20 13:38:21 ragnarok sshd[17140]: pam_unix(sshd:session): session closed for user
Jul 20 13:47:02 ragnarok systemd[1]: Stopping OpenBSD Secure Shell server...
Jul 20 13:47:02 ragnarok sshd[16883]: Received signal 15; terminating.
Jul 20 13:47:02 ragnarok systemd[1]: ssh.service: Deactivated successfully.
Jul 20 13:47:02 ragnarok systemd[1]: Stopped OpenBSD Secure Shell server.
-- Boot ffa2b75f79a04f4196cfc15a8c8c245e --
Jul 20 13:47:24 ragnarok systemd[1]: Starting OpenBSD Secure Shell server...
Jul 20 13:47:24 ragnarok sshd[913]: Server listening on 0.0.0.0 port 22.
Jul 20 13:47:24 ragnarok sshd[913]: Server listening on :: port 22.
Jul 20 13:47:24 ragnarok systemd[1]: Started OpenBSD Secure Shell server.
Jul 20 15:48:31 ragnarok sshd[913]: Received signal 15; terminating.
Jul 20 15:48:31 ragnarok systemd[1]: Stopping OpenBSD Secure Shell server...
Jul 20 15:48:31 ragnarok systemd[1]: ssh.service: Deactivated successfully.
Jul 20 15:48:31 ragnarok systemd[1]: Stopped OpenBSD Secure Shell server.
-- Boot 3a3ca0eb171e4af2ba2fca7a175f582d --
Jul 23 13:51:18 ragnarok systemd[1]: Starting OpenBSD Secure Shell server...

```

Figure 22.1: Viewing logging information via the journalctl command

The `-u` option is required and tells the `journalctl` command that you'd like to check on a service. So in the previous example, we provided `ssh` as an argument for what we wanted to find logging information on. The name of the unit (or service) will be the same as it is when you're starting, stopping, or restarting a service with the `systemctl` command. I recommend making the `journalctl` command your first consideration when checking logging information on a Linux system.



If you also add the `-f` option in addition to the `-u` option, the output will continue to scroll as new information is added to the logs for the particular service you're checking. This is very useful if you want to follow along with what's going on with a service as new events occur.

However, not all services log information via `journalctl`, so understanding the legacy approach to viewing log files is also important. Inside the `/var/log` directory, you'll see a handful of logs you can view, which differs from server to server depending on which applications are installed. In quite a few cases, an installed application will create its own log file somewhere within `/var/log`, either in a log file or a log file within a subdirectory of `/var/log`. For example, once you install Apache, it will create log files in the `/var/log/apache2` directory, and looking through those logs may give you a hint as to what may be going on if the problem is related to your web server. These are known as **Application Logs**, which are basically log files created by an application and not the distribution. There are also **System Logs**, which are the log files created by the distribution and allow you to view system events.

Viewing a log file stored within the `/var/log` directory can be done in several ways. One way is to use the `cat` command along with the path and filename of a log file. For example, the Apache access log can be viewed with the following command:

```
cat /var/log/apache2/access.log
```

Some log files are restricted and need root privileges in order to access them. If you get a permission denied error when attempting to view a log, use `sudo` in front of any of the commands in this section to view the file.

One problem with the `cat` command is that it will print out the entire file, no matter how big it is. It will scroll by your terminal and if the file is large, you won't be able to see all of it. In addition, if your server is already taxed when it comes to performance, using `cat` can actually tie up the server for a bit in a case where the log file is massive. This will cause you to lose control of your shell until the file stops printing. You can press `Ctrl + c` to stop printing the log file, but the server may end up being too busy to respond to `Ctrl + c` and show the entire file anyway.

Another method is to use the `tail` command. By default, the `tail` command shows you the last ten lines of a file:

```
tail /var/log/apache2/access.log
```

If you wish to see more than the last ten lines, you can use the `-n` option to specify a different amount. To view the last 100 lines, we would use the following:

```
tail -n 100 /var/log/apache2/access.log
```

Perhaps one of the most useful features of the `tail` command is the `-f` option, which allows you to follow a log file. Basically, this means that as entries are written to the log file, it will scroll by in front of you. It's close to watching the log file in real time:

```
tail -f /var/log/apache2/access.log
```

Once you start using the follow option, you'll wonder how you ever lived without it. If you're having a specific problem that you are able to reproduce, you can watch the log file for that application and see the log entries as they appear while you're reproducing the issue. In the case of a DHCP server not providing IP addresses to clients, you can view the output of the `/var/log/syslog` file (the `isc-dhcp-server` daemon doesn't have its own log file), and you can see any errors that come up as your clients try to re-establish their DHCP lease, allowing you to see the problem as it is happening.

Another useful command for viewing logs is `less`. The `less` command allows you to scroll through a log file with the page up and page down keys on your keyboard, which makes it more useful for viewing log files than the `cat` command. You can press `q` to exit the file:

```
less /var/log/apache2/access.log
```

So now that you know a few ways in which you can view these files, which files should you inspect? Unfortunately, there's no one rule, as each application handles its logging differently. Some daemons have their own log file stored somewhere in `/var/log`.

Therefore, a good place to check is in that directory, to see if there is a log file with the name of the daemon. Some daemons don't even have their own log file and will use `/var/log/syslog` instead. You may try viewing the contents of the file while using `grep` to find messages related to the daemon you're troubleshooting. In regard to the `isc-dhcp-server` daemon, the following would narrow down the `syslog` to messages from that specific daemon:

```
cat /var/log/syslog | grep dhcp
```

While troubleshooting security issues, the log file you'll definitely want to look at is the **Authorization Log**, located at `/var/log/auth.log`. You'll need to use the root account or `sudo` to view this file. The authorization log includes information regarding authentication attempts to the server, including logins from the server itself, as well as logins over OpenSSH. This is useful for several reasons, among them the fact that if something really bad happens on your server, you can find out who logged in to the server around that time. In addition, if you or one of your users is having trouble accessing the server via OpenSSH, you may want to look at the authorization log for clues, as additional information for OpenSSH failures will be logged there. Often, the `ssh` command may complain about permissions of key files not being correct, which would give you an answer as to why public key authentication stopped working, as OpenSSH expects specific permissions for its files. For example, the private key file (typically `/home/<user>/.ssh/id_rsa`) should not be readable or writable by anyone other than its owning user. You'd see errors within `/var/log/auth.log` mentioning such a thing if that were the case.

Another use case for checking `/var/log/auth.log` is for security, as a high number of login attempts may indicate an intrusion attempt. (Hopefully, you have Fail2ban installed, which we went over in the last chapter.) An unusually high number of failed password attempts may indicate someone trying to log in to the server by brute force. That would definitely be a cause for concern, and you'd want to block their IP address immediately.

The **System Log**, located in `/var/log/syslog`, contains logging information for quite a few different things. It's essentially the Swiss Army knife of Ubuntu's logs. If a daemon doesn't have its own log file, chances are its logs are being written to this file. In addition, information regarding cron jobs will be written here, which makes it a candidate to check when a cron job isn't being executed properly. The `dhclient` daemon, which is responsible for grabbing an IP address from a DHCP server, is also important.

You'll be able to see from `dhclient` events within the system log when an IP address is renewed, and you can also see messages relating to failures if it's not able to obtain an IP address. Also, the `systemd` `init` daemon itself logs here, which allows you to see messages related to server startup as well as applications it's trying to run.

Another useful log is the `/var/log/dpkg.log` file, which records log entries relating to installing and upgrading packages. If a server starts misbehaving after you roll out updates across your network, you can view this log to see which packages were recently updated. This log will not only give you a list of updated or installed packages, but also a timestamp from when the installation occurred. If a user installed an unauthorized application, you can correlate this log to the authentication log to determine who logged in around that time, and then you can check that user's Bash history to confirm.

Often, log files will get rotated after some time by a utility known as `logrotate`. Inside the `/var/log` directory, you'll see several log files with a `.gz` extension, which means that the original log file was compressed and renamed, and a new log file was created in its place. For example, you'll see the `syslog` file for the system log in the `/var/log` directory, but you'll also see files named with a number and a `.gz` extension as well, such as `syslog.2.gz`. These are compressed logs. Normally, you'd view these logs by uncompressing them and then opening them via any of the methods mentioned in this section. An easier way to do so is with the `zcat` command, which allows you to view compressed files immediately:

```
zcat /var/log/syslog.2.gz
```

There's also `zless`, which serves a similar purpose as the `less` command.

Another useful command for checking logging information is `dmesg`. Unlike other log files, `dmesg` is literally its own command, and you can execute it from anywhere in the filesystem. The `dmesg` command allows you to view log entries from the Linux kernel's ring buffer, which can be very useful when troubleshooting hardware issues (such as seeing which disks were recognized by the kernel). When troubleshooting hardware, the system log is also helpful, but using the `dmesg` command may be a good place to check as well.

As I mentioned earlier, on an Ubuntu system, there are two types of log files, system logs and application logs. System logs, such as `auth.log` and `dpkg.log`, detail important system events and aren't specific to any one particular application. Application logs become installed when you install their parent packages, such as Apache or MariaDB. Application logs create log entries into their own log file.

Some daemons you install will not create their own application log, such as `isc-dhcp-server`. Since there's no general rule when it comes to which applications log is where, the first step in finding a log file is to see if the application you want log entries from creates its own log file. If not, it's likely using a system log.

When faced with a problem, it's important to practice viewing log files at the same time as you try and reproduce the problem. Using follow mode with `tail` (`tail -f`) works very well for this, as you can watch the log file generate new entries as you try and reproduce the issue. This technique works very well in almost any situation where you're dealing with a misbehaving daemon. This technique can also help narrow down hardware issues. For example, I once dealt with an Ubuntu system where when I plugged in a flash drive, nothing happened. When I followed the log as I inserted and removed the flash drive, I saw the system log update and recognize each insertion and removal. So clearly, the Linux kernel itself saw the hardware and was prepared to use it. This helped me narrow down the problem to being that the desktop environment I was using wasn't updating to show the inserted flash drive, but my hardware and USB ports were operating perfectly fine. With one command, I was able to determine that the issue was a software problem and not related to hardware.

As you can see, Ubuntu contains very helpful log files that will aid you in troubleshooting your servers. Often, when you're faced with a problem, viewing relevant log entries and then conducting a Google search regarding them will result in a useful answer, or at least bring you to a bug report to let you know the problem isn't just limited to you or your configuration.

Hopefully, your search results will lead you right to the answer, or at least to a workaround. From there, you can continue to work through the problem until it is solved.

What about network issues? Tracking down the root cause of an issue on the network can be especially challenging, but it's not as difficult as it may seem. In the next section, we'll take a look at a few ways you can trace network issues.

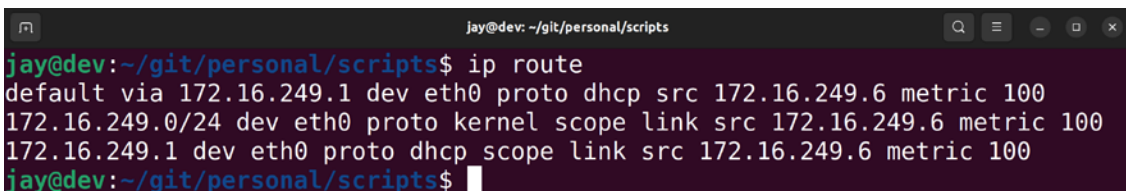
Tracing network issues

It's amazing how important TCP/IP networking is to the world today. Of all the protocols in use in modern computing, it's by far the most widespread. But it's also one of the most annoying situations to figure out when it's not working well. Thankfully, Ubuntu features really handy utilities you can use in order to pinpoint what's going on.

First, let's look at connectivity. After all, if you can't connect to a network, your server is essentially useless. In most cases, Ubuntu recognizes just about all network cards without fail, and it will automatically connect your server or workstation to your network if it is within reach of a DHCP server.

While troubleshooting, get the obvious stuff out of the way first. The following may seem like a no-brainer, but you'd be surprised how often one can miss something obvious. I'm going to assume you've already checked to make sure network cables are plugged in tight on both ends. Another aspect regarding cabling is that sometimes network cables themselves develop faults and need to be replaced. You should be able to use a cable tester and get a clean signal through the cable.

Routing issues can sometimes be tricky to troubleshoot, but by testing each destination point one by one, you can generally see where the problem lies. Typical symptoms of a routing issue may include being unable to access a device within another subnet, or perhaps not being able to get out to the internet, despite being able to reach internal devices. To investigate a potential routing issue, first, check your routing table. You can do so with the `ip route` command. This command will print your current routing table information:

A terminal window with a dark background and light text. The prompt is 'jay@dev:~/git/personal/scripts\$'. The command 'ip route' has been entered and executed. The output consists of three lines of text: 'default via 172.16.249.1 dev eth0 proto dhcp src 172.16.249.6 metric 100', '172.16.249.0/24 dev eth0 proto kernel scope link src 172.16.249.6 metric 100', and '172.16.249.1 dev eth0 proto dhcp scope link src 172.16.249.6 metric 100'. The prompt is now 'jay@dev:~/git/personal/scripts\$' with a cursor at the end.

```
jay@dev:~/git/personal/scripts$ ip route
default via 172.16.249.1 dev eth0 proto dhcp src 172.16.249.6 metric 100
172.16.249.0/24 dev eth0 proto kernel scope link src 172.16.249.6 metric 100
172.16.249.1 dev eth0 proto dhcp scope link src 172.16.249.6 metric 100
jay@dev:~/git/personal/scripts$
```

Figure 22.2: Viewing the routing table on an Ubuntu server

In this example, you can see that the default gateway for all traffic is `10.10.10.1`. This is the first entry on the table, which tells us that all traffic to the destination `0.0.0.0` (which is everything) leaves via `10.10.10.1`. As long as ICMP traffic isn't disabled, you should be able to ping this default gateway, and you should be able to ping other nodes within your subnet as well.

To start troubleshooting a routing issue, you would use the information shown after printing your routing table to conduct several ping tests. First, try to ping your default gateway. If you cannot, then you've found the issue. If you can, try running the `traceroute` command.

This command isn't available by default, but all you'll have to do is install the `traceroute` package, so hopefully, you have it installed on the server. If you do, you can run `traceroute` against a host, such as an external URL, to find out where the connection drops. The `traceroute` command should show every hop between you and your target. Each "hop" is basically another default gateway. You traverse through one gateway after another until you ultimately reach your destination. With the `traceroute` command, you can see where the chain stops. In all likelihood, you'll find that perhaps the problem isn't even on your network, but perhaps your internet service provider is where the connection drops.

DNS issues don't happen very often, but by using a few tricks, you should be able to resolve them. Symptoms of DNS failures will usually result in a host being unable to access internal or external resources by name.

Knowing whether the problem is with internal or external hosts (or both) should help you determine whether it's your DNS server that's the problem, or perhaps the DNS server at your ISP.

The first step in pinpointing the source of DNS woes is to ping a known IP address on your network, preferably the default gateway. If you can ping it, but you can't ping the gateway by name, then you probably have a DNS issue. You can confirm a potential DNS issue by using the `nslookup` command against the domain, such as this:

```
nslookup myserver.local
```

In addition, make sure you try and ping external resources as well, such as a website. This will help you narrow down the scope of the issue.

You will also want to know which DNS server your host is sending queries to. In the past, finding out which DNS server was assigned to your host was as simple as inspecting the contents of `/etc/resolv.conf`. However, nowadays, this file will often refer to a local resolver instead and won't reveal the actual server requests are being sent to. To find out the real DNS server that's assigned to your host, the following command will do the trick:

```
resolvectl status
```

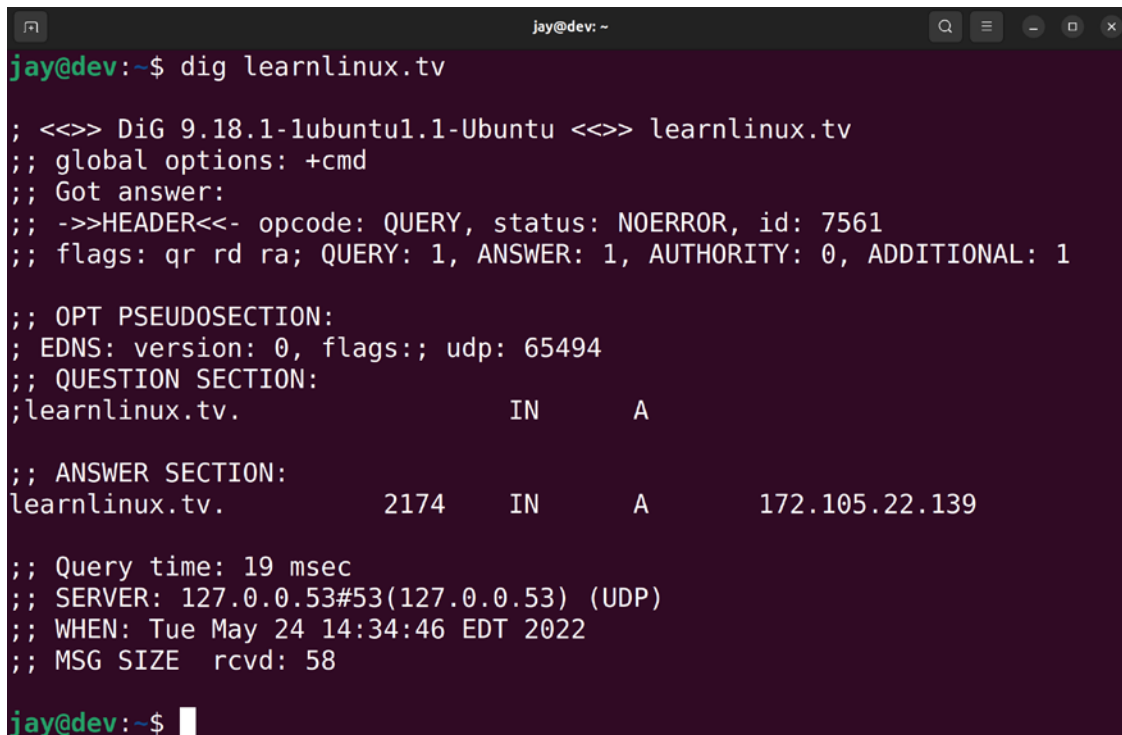
Are they what you expect? If not, you can temporarily fix this problem by removing the incorrect name server entries from this file and replacing them with the correct IP addresses. The reason I suggest this as a temporary fix and not a permanent one is because the next thing you'll need to do is investigate how the invalid IP addresses got there in the first place. Normally, these are assigned by your DHCP server. As long as your DHCP server is sending out the appropriate name server list, you shouldn't run into this problem. If you're using a static IP address, then perhaps there's an error in your Netplan config file.

A useful method of pinpointing DNS issues in regard to being unable to resolve external sites is to temporarily switch your DNS provider on your local machine. Normally, your machine is going to use your external DNS provider, such as the one that comes from your ISP. Your external DNS server is something we went through setting up in *Chapter 11, Setting Up Network Services*, specifically the `forwarders` section of the configuration for the `bind9` daemon. The `forwarders` used by the `bind9` daemon are where it sends traffic if it isn't able to resolve your request based on its internal list of hosts.

You could consider bypassing this by changing your local workstation's DNS name servers to Google's, which are 8.8.8.8 and 8.8.4.4. If you're able to reach the external resource after switching your name servers, you can be reasonably confident that your forwarders are the culprit.

I've actually seen situations in which a website has changed its IP address but the ISP's DNS servers didn't get updated quickly enough, causing some clients to be unable to reach a site they need to perform their job. Switching everyone to alternate name servers (by adjusting the forwarders option, as we did in *Chapter 11, Setting Up Network Services*) was the easiest way they could work around the issue.

Some additional tools to consider while checking your server's ability to resolve DNS entries are `dig` and `nslookup`. You should be able to use both commands to test your server's DNS settings. Both commands are used with a hostname or domain name as an option. The `dig` command will present you with information regarding the address (A) record of the DNS zone file responsible for the IP address or domain. The `host` command should return the IP address of the host you're trying to reach. Here's some example output:

A terminal window with a dark background and light text. The prompt is 'jay@dev:~\$'. The command 'dig learnlinux.tv' has been entered. The output shows DNS query details: 'DiG 9.18.1-lubuntu1.1-Ubuntu <<>> learnlinux.tv', global options '+cmd', and 'Got answer:'. The header section shows 'opcode: QUERY, status: NOERROR, id: 7561' and 'flags: qr rd ra; QUERY: 1, ANSWER: 1, AUTHORITY: 0, ADDITIONAL: 1'. The question section shows 'learnlinux.tv. IN A'. The answer section shows 'learnlinux.tv. 2174 IN A 172.105.22.139'. The query time is 19 msec, the server is 127.0.0.53#53, and the message size received is 58 bytes.

```
jay@dev:~$ dig learnlinux.tv
; <<>> DiG 9.18.1-lubuntu1.1-Ubuntu <<>> learnlinux.tv
;; global options: +cmd
;; Got answer:
;; ->>HEADER<<- opcode: QUERY, status: NOERROR, id: 7561
;; flags: qr rd ra; QUERY: 1, ANSWER: 1, AUTHORITY: 0, ADDITIONAL: 1

;; OPT PSEUDOSECTION:
; EDNS: version: 0, flags;; udp: 65494
;; QUESTION SECTION:
;learnlinux.tv.                IN      A

;; ANSWER SECTION:
learnlinux.tv.                2174    IN      A      172.105.22.139

;; Query time: 19 msec
;; SERVER: 127.0.0.53#53(127.0.0.53) (UDP)
;; WHEN: Tue May 24 14:34:46 EDT 2022
;; MSG SIZE rcvd: 58

jay@dev:~$
```

Figure 22.3: Output of the `dig` and `host` commands

Hardware support is also critical when it comes to networking. If the Linux kernel doesn't support your network hardware, then you'll likely run into a situation where the distribution doesn't recognize or do anything when you insert a network cable, or in the case of wireless networking, doesn't show any nearby networks despite there being one or more. Unlike the Windows platform, hardware support is generally baked right into the kernel when it comes to Linux. While there are exceptions to this, the Linux kernel shipped with a distribution typically supports hardware the same age as itself or older.

In the case of Ubuntu 22.04 LTS (which was released in April 2022), it's able to support hardware released as of the beginning of 2022 and older. Future releases of Ubuntu Server will publish hardware enablement updates, which will allow Ubuntu Server 22.04 to support newer hardware and chip sets once they come out. Typically, Ubuntu will release several point releases during the life of a supported distribution, such as 22.04.1, 22.04.2, and so on. As long as you're using the latest one, you'll have access to the latest hardware support that Ubuntu has made available at the time.

In other cases, hardware support may depend on external kernel modules. In the case of a missing hardware driver, the first thing you should try when faced with network hardware that's not recognized is to look up the hardware using a search engine. Typically the search term `<hardware name> Ubuntu` will do the trick. But what do you search for? To find out the hardware string for your network device, try the `lspci` command:

```
lspci | grep -i net
```

The `lspci` command lists hardware connected to your server's PCI bus. Here, we're using the command with a case insensitive `grep` search for the word `net`:

```
lspci |grep -i net
```

This should return a list of networking components available on your server. On my machine, for example, I get the following output:

```
01:00.1 Ethernet controller: Realtek Semiconductor Co., Ltd. RTL8111/8168/8411
PCI Express Gigabit Ethernet Controller (rev 12)
02:00.0 Network controller: Intel Corporation Wireless 8260 (rev 3a)
```

As you can see, I have a wired and wireless network card on this machine. If one of them wasn't working, I could search online for information by searching for the hardware string and the keyword `Ubuntu`, which should give me results pertaining to my exact hardware. If a package is required to be installed, the search results will likely give me some clues as to what package I need to install.

Without having network access though, the worst-case scenario is that I may have to download the package from another computer and transfer it to the server via a flash drive. That's certainly not a fun thing to need to do, but it does work if the latest Ubuntu installation media doesn't yet offer full support for your hardware.

Another potential problem point is DHCP. When it works well, DHCP is a wonderfully magical thing. When it stops working, it can be frustrating. But generally, DHCP issues often end up being a lack of available IP addresses, the DHCP daemon (`isc-dhcp-server`) not running, an invalid configuration, or hosts that have clocks that are out of sync (all servers should have the `ntp` package installed).

If you have a server that is unable to obtain an IP address via DHCP and your network utilizes a Linux-based DHCP server, check the system log (`/var/log/syslog`) for events related to `dhcpcd`. Unfortunately, there's no command you can run that I've ever been able to find that will print how many IP address leases your DHCP server has remaining, but if you run out, chances are you'll see log entries related to an exhausted pool in the system log. In addition, the system log will also show you attempts from your nodes to obtain an IP address as they attempt to do so. Feel free to use `tail -f` against the system log to watch for any events related to DHCP leases.

In some cases, a lack of DHCP leases being available can come down to having a very generous lease time enabled. Some administrators will give their clients up to a week for the lease time, which is generally unnecessary. A lease time of one day is fine for most networks, but ultimately, the lease time you decide on is up to you. In *Chapter 11, Setting Up Network Services*, we looked at configuring our DHCP server, so feel free to refer to that chapter if you need a refresher on how to configure the `isc-dhcp-server` daemon.

Although it's probably not the first thing you'll think of while facing DHCP issues, hosts having out-of-sync clocks can actually contribute to the problem. DHCP requests are timestamped on both the client and the server, so if the clock is off by a large degree on one, the timestamps will be off as well, causing the DHCP server to become confused. Surprisingly, I've seen this come up fairly often. I recommend standardizing NTP across your network as early on as you can. DHCP isn't the only service that suffers when clocks are out of sync; file synchronization utilities also require accurate time. If you ensure NTP is installed on all of your clients and it's up to date and working, you should be in good shape. Using configuration management utilities such as Ansible to ensure NTP is not only configured but is running properly on all the machines in your network will only benefit you.

Of course, there are many things that can go wrong when it comes to networking, but the information here should cover the majority of issues. In summary, troubleshooting network issues generally revolves around ping tests. Trying to ping your default gateway, tracing failed endpoints with `traceroute`, and troubleshooting DNS and DHCP will take care of a majority of issues. Then again, faulty hardware such as failed network cards and bad cabling will no doubt present themselves as well.

Our servers utilize storage, CPU, memory, and other resources to provide us with value and serve our clients. In the next section, we'll take a closer look at how to check those resources.

Troubleshooting resource issues

I don't know about others, but it seems that a majority of my time troubleshooting servers is usually spent pinpointing resource issues. By resources, I'm referring to CPU, memory, disk, input/output, and so on. Generally, issues come down to a user storing too many large files, a process going haywire that consumes a large amount of CPU, or a server running out of memory. In this section, we'll go through some of the common things you're likely to run into while administering Ubuntu servers.

First, let's revisit topics related to storage. In *Chapter 9, Managing Storage Volumes*, we went over concepts related to this already, and many of those concepts also apply to troubleshooting as well. Therefore, I won't spend too much time on those concepts here, but it's worth a refresher in regard to troubleshooting storage issues. First, whenever you have users that are complaining about being unable to write new files to the server, the following two commands are the first you should run. You are probably already well aware of these, but they're worth repeating:

```
df -h
df -i
```

The first `df` command variation gives you information regarding how much space is used on a drive, in a human-readable format (the `-h` option), which will print the information in terms of megabytes and gigabytes.

The `-i` option in the second command gives you information regarding in-use and available inodes. The reason you should also run this is that on a Linux system, it can report storage as full even if there's plenty of free space. But if there are no remaining inodes, it's the same as being full, but the first command wouldn't show the usage as 100 percent when no inodes are free. Usually, the number of inodes a storage medium has available is extremely generous, and the limit is hard to hit. However, if a service is creating new log files over and over every second, or a mail daemon grows out of control and generates a huge backlog of undelivered mail, you'd be surprised how quickly inodes can empty out.

Of course, once you figure out that you have an issue with full storage, the next logical question becomes, what is eating up all my free space? The `df` commands will give you a list of storage volumes and their sizes, which will tell you at least which disk or partition to focus your attention on. My favorite command for pinpointing storage hogs, as I mentioned in *Chapter 9, Managing Storage Volumes*, is the `ncdu` command. While not installed by default, `ncdu` is a wonderful utility for checking to see where your storage is being consumed the most. If run by itself, `ncdu` will scan your server's entire filesystem. Instead, I recommend running it with the `-x` option, which will limit it to a specific folder as a starting point. For example, if the `/home` partition is full on your server, you might want to run the following to find out which directory is using the most space:

```
sudo ncdu -x /home
```

The `-x` option will cause `ncdu` to not cross filesystems. This means if you have another disk mounted within the folder you're scanning, it won't touch it. With `-x`, `ncdu` is only concerned with the target you give it.

If you aren't able to utilize `ncdu`, there's also the `du` command, which takes some extra work. The `du -h` command, for example, will give you the current usage of your current working directory, with human-readable numbers. It doesn't traverse directory trees by default like `ncdu` does, so you'd need to run it on each subdirectory until you manually find the directory that's holding the most files. A very useful variation of the `du` command, nicknamed `ducks`, is the following. It will show you the top 15 largest directories in your current working directory:

```
du -cksh * | sort -hr | head -n 15
```

Another issue with storage volumes that can arise is to do with filesystem integrity. Most of the time, these issues only seem to come up when there's an issue with power, such as a server powering off unexpectedly. Depending on the server and the formatting you've used when setting up your storage volumes (and several other factors), power issues are handled differently from one installation to another. In most cases, a filesystem check (`fsck`) will happen automatically during the next boot. If it doesn't, and you're having odd issues with storage that can't be explained, a manual filesystem check is recommended. Scheduling a filesystem check is actually very easy:

```
sudo touch /forcefsck
```

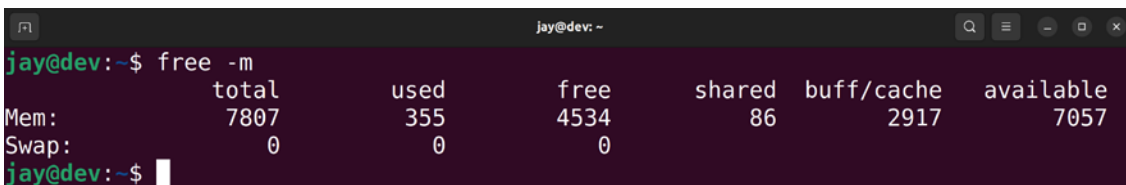
The previous command will create an empty file, `forcefsck`, at the root of the filesystem. When the server reboots and it sees this file, it will trigger a filesystem check on that volume and then remove the file.

If you'd like to check a filesystem other than the root volume, you can create the `forcefsck` file elsewhere. For example, if your server has a separate `/home` partition, you could create the file there instead to check that volume:

```
sudo touch /home/forcefsck
```

The filesystem check will usually complete fairly quickly unless there's an issue it needs to fix. Depending on the nature of the problem, the issue could be repaired quickly, or perhaps it will take a while. I've seen some really bad integrity issues that have taken over four hours to fix, but I've seen others fixed in a matter of seconds. Sometimes it will finish so quickly that it will scroll by so fast during boot that you may miss seeing it. In case of a large volume, you may want to schedule the `fsck` check to happen after-hours in case the scan takes a long time.

With regards to issues with memory, the `free -m` command will give you an overview of how much memory and swap are available on your server. It won't tell you what exactly is using up all your memory, but you'll use it to see if you're in jeopardy of running out. The **free** column from the output of the `free` command will show you how much memory is remaining, and allow you to make a decision on when to take action:

A terminal window titled 'jay@dev: -' showing the output of the 'free -m' command. The output is a table with columns: total, used, free, shared, buff/cache, and available. The rows are Mem: and Swap:.

	total	used	free	shared	buff/cache	available
Mem:	7807	355	4534	86	2917	7057
Swap:	0	0	0			

Figure 22.4: Checking the available memory on an Ubuntu server

In *Chapter 8, Monitoring System Resources*, we took a look at the `htop` command, which helps us answer the question of “what” is using up our resources. Using `htop` (once installed), you can sort the list of processes by CPU or memory usage by pressing `F6` and then selecting a new sort field, such as `PERCENT_CPU` or `PERCENT_MEM`. This will give you an idea of what is consuming resources on your server, allowing you to make a decision on what to do about it. The action you take will differ from one process to another, and your solution may range from adding more memory to the server to tuning the application to have a lower memory ceiling. But what do you do when the results from `htop` don't correlate to the usage you're seeing? For example, what if your load average is high, but no process seems to be consuming a large portion of CPU?

One command I haven't discussed so far in this book is `iostat`. While not installed by default, the `iostat` utility is definitely a must-have, so I recommend you install the `iostat` package. The `iostat` utility itself needs to be run as root or with `sudo`:

```
sudo iostat
```

The `iostat` command will allow you to see how much data is being written to or read from your disks. **Input/Output (IO)** definitely contributes to a system's load, and not all resource monitoring utilities will show this usage. If you see a high load average but nothing in your resource monitor shows anything to account for it, check the IO.

The `iostat` utility is a great way to do that as if data is bottle-necked while being written to disk, which can account for a serious overhead in IO that will slow other processes down. If nothing else, it will give you an idea of which process is misbehaving, in case you need to kill it:

```

Total DISK READ:      0.00 B/s | Total DISK WRITE:      0.00 B/s
Current DISK READ:    0.00 B/s | Current DISK WRITE:    0.00 B/s
  TID  Prio  USER   DISK READ  DISK WRITE  SWAPIN     IO>   COMMAND
  1 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  systemd f~rialize 40
  2 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [kthreadd]
  3 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [rcu_gp]
  4 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [rcu_par_gp]
  8 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [mm_percpu_wq]
  9 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [rcu_tasks_kthre]
 10 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [rcu_tasks_rude_]
 11 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [rcu_tasks_trace]
 12 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [ksoftirqd/0]
 13 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [rcu_preempt]
 14 ?sys  root    0.00 B/s   0.00 B/s   ?unavailable?  [migration/0]
keys:  any: refresh  q: quit  i: ionice  o: active  p: procs  a: accum
sort:  r: asc  left: SWAPIN  right: COMMAND  home: TID  end: COMMAND
CONFIG_TASK_DELAY_ACCT not enabled in kernel, cannot determine SWAPIN and IO %

```

Figure 22.5: The `iotop` utility running on an Ubuntu server

The `iotop` window will refresh on its own, sorting processes by the column that is highlighted. To change the highlight, you'll only need to press the *left* and *right* arrows on your keyboard. You can sort processes by columns such as IO, SWAPIN, DISK WRITE, DISK READ, and others. When you're finished with the application, press `q` to quit.

The utilities we looked at in this section are very useful when identifying issues with bottle-necked resources. What you do to correct the situation after you find the culprit will depend on the daemon. Perhaps there's an invalid configuration, or the daemon has encountered a fault and needs to be restarted. Often, checking the logs may lead you to an answer as to why a daemon misbehaves. In the case of full storage, almost nothing beats `ncdu`, which will almost always lead you directly to the problem. Tools such as `htop` and `iotop` allow you to view additional information regarding resource usage as well, and `htop` even allows you to kill a misbehaving process right from within the application, by pressing `F9`.

What do you do when system memory (RAM) becomes physically defective? It happens more often than you'd think. In the next section, we'll look at a way we can test our RAM to see if it's defective or not.

Diagnosing defective RAM

All server and computing components can and will fail eventually, but there are a few pieces of hardware that seem to fail more often than others. Fans, power supplies, and hard disks definitely make the list of common things administrators will end up replacing, but defective memory is also a situation I'm sure you'll run into eventually.

Although memory sticks becoming defective is something that could happen, I made it the last section in this chapter because unfortunately, I can't give you a definitive list of symptoms to look out for that indicate that memory is the source of an issue. RAM issues are very mysterious in nature, and each time I've run into one, I've always stumbled across memory being bad only after troubleshooting everything else. It's for this reason that nowadays I'll often test the memory on a server or workstation first since it's very easy to do. Even if memory has nothing to do with an issue, it's worth checking anyway since it could become a problem later.

Most distributions of Linux (Ubuntu included) feature **Memtest86+** right on the installation media. Whether you create a bootable CD or flash drive, there's a memory test option available from the Ubuntu Server media. When you first boot from the Ubuntu Server media, you'll see an icon toward the bottom indicating that you can press a key to bring up a menu (if you don't press a key, the installer will automatically start). Next, you'll be asked to choose your language, and then you'll be shown an installation menu. Among the choices there will be a **Test memory** option:

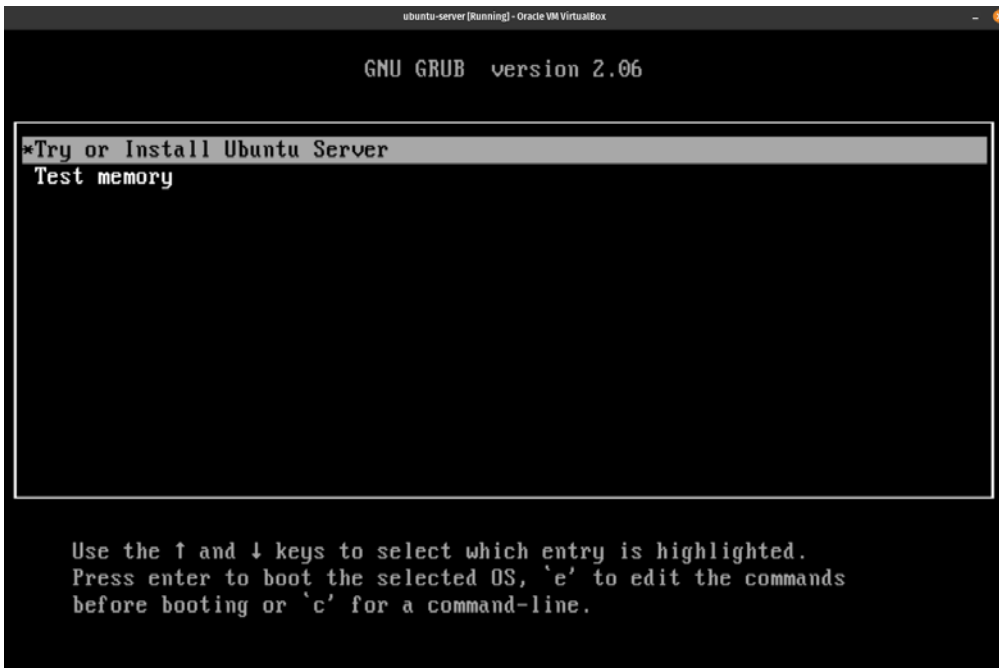


Figure 22.6: The main menu of the Ubuntu installer, showing a *Test memory* option

Other editions of Ubuntu, such as the Ubuntu desktop distribution or any of its derivatives, also feature an option to test memory. Even if you don't have installation media handy for the server edition, you can use whichever version you have.

When you choose the **Test memory** option from your installation media, the **Memtest86+** program will immediately get to work and start testing your memory (press *Esc* to exit the test). The test may take a long time, depending on how much memory your workstation or server has installed. It can take minutes or even hours to complete. Generally speaking, when your machine has defective RAM, you'll see a bunch of errors show up relatively quickly, usually within the first 5-10 minutes.

If you don't see errors within 15 minutes, you're most likely in good shape. In my experience, every time I've run into defective memory, I've seen errors in 15 minutes or less (usually within 5). Theoretically, though, you could very well have a small issue with your memory modules that may not show up until after 15 minutes, so you should let the test finish if you can spare the time for it.

The main question becomes when to run **Memtest86+** on a machine. In my experience, symptoms of bad memory are almost never the same from one machine to another. Usually, you'll run into a situation where a server doesn't boot properly, applications close unexpectedly, applications don't start at all, or perhaps an application is behaving irregularly. In my view, testing memory should be done whenever you experience a problem that doesn't necessarily seem straightforward. In addition, you may want to consider testing the memory on your server before you roll it out into production. That way, you can ensure that it starts out as free of hardware issues as possible. If you install new memory modules, make sure to test the RAM right away.

If the test does report errors, you'll next want to find out which memory module is faulty. This can be difficult, as some servers can have more than a dozen memory modules installed. To narrow it down, you'd want to test each memory module independently if you can, until you find out which one is defective. You should also continue to test the other modules, even after you discover the culprit. The reason for this is that having multiple memory modules going bad isn't outside the realm of possibility, considering whatever situation led to the first module becoming defective may have affected others.

Another tip I'd like to pass along regarding memory is that when you do discover a bad stick of memory, it's best to erase the hard disk and start over if you can. I understand that this isn't always feasible, and you could have many hours logged into setting up a server. Some servers can take weeks to rebuild, depending on their workload. But at least keep in mind that any data that passes through defective RAM can become corrupted.

This means that data at rest (data stored on your hard disk) may be corrupt if it was sitting in a defective area of RAM before it was written to disk. When a server or workstation encounters defective RAM, you really can't trust it anymore. I'll leave the decision on how to handle this situation up to you (hopefully you'll never encounter it at all), but just keep this in mind as you plan your course of action. Personally, I don't trust an installation of any operating system after its hardware has encountered such issues.

I also recommend that you check the capacitors on your server's motherboard whenever you're having odd issues. Although this isn't necessarily related to memory, I mention it here because the symptoms are basically the same as bad memory when you have bad capacitors. I'm not asking you to get a voltage meter or do any kind of electrician work, but sometimes it may make sense to open the case of your server, shine a flashlight on the capacitors, and see if any of them appear to be leaking fluid or expanding. The reason I bring this up is that I've personally spent hours troubleshooting a machine (more than once) where I would test the memory and hard disk and look through system logs, without finding any obvious causes, only to later look at the hardware and discover that capacitors on the motherboard were leaking. It would have saved me a lot of time if I had simply looked at the capacitors. And that's really all you have to do: just take a quick glance around the motherboard and look for anything that doesn't seem right.

Summary

While Ubuntu is generally a very stable and secure platform, it's important to be prepared for problems occurring and to know how to deal with them. In this chapter, we discussed common troubleshooting we can perform when our servers stop behaving themselves. We started off by evaluating the scope, which gives us an understanding of how many users or servers are affected by the issue. Then, we looked into Ubuntu's log files, which are a treasure trove of information that we can use to pinpoint issues and narrow down the problem. We also covered several networking issues that can come up, such as issues with DHCP, DNS, and routing. We certainly can't predict problems before they occur, nor can we be prepared in advance for every type of problem that can possibly happen. However, applying sound logic and common sense to problems will go a long way in helping us figure out the root cause.

In the next chapter, we will take a look at preventing disasters in the first place and recovering from them if they happen anyway. See you there!

Further reading

- Reporting Bugs in Ubuntu Server: <https://learnlinux.link/report-bugs>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>



23

Preventing Disasters

In an enterprise network, a disaster can strike at any time. While, as administrators, we always do our best to design the most stable and fault-tolerant server implementations we possibly can, what matters most is how we are able to deal with disasters when they do happen. As stable as server hardware is, any component of a server can fail at any time. In the face of a disaster, we need a plan. How can you attempt to recover data from a failed disk? What do you do when your server all of a sudden decides it doesn't want to boot? These are just some of the questions we'll answer as we take a look at several ways we can prevent and recover from disasters. In our final chapter, we'll cover the following topics:

- Preventing disasters
- Utilizing Git for configuration management
- Implementing a backup plan
- Utilizing bootable recovery media

We'll start off our final chapter by looking at a few tips to help prevent disaster.

Preventing disasters

As we proceed through this chapter, we'll look at ways we can recover from disasters. However, if we can prevent a disaster from occurring in the first place, then that's even better. We certainly can't prevent every type of disaster that could possibly happen but having a good plan in place and following that plan will lessen the likelihood. A good disaster recovery plan will include a list of guidelines to be followed with regard to implementing new servers and managing current ones.

This plan may include information such as an approved list of hardware (such as hardware configurations known to work efficiently in an environment), as well as rules and regulations for users, a list of guidelines to ensure physical and software security, proper training for end users, and method change control. Some of these concepts we've touched on earlier in the book but are worth repeating from the standpoint of disaster prevention.

First, we talked about the **principle of least privilege** back in *Chapter 21, Securing Your Server*. The idea is to give your users as few permissions as possible. This is very important for security, as you want to ensure only those trained in their specific jobs are able to access and modify only the resources that they are required to. Accidental data deletion happens all the time. To take full advantage of this principle, create a set of groups as part of your overall security design. List departments and positions in your company and the types of activities each is required to perform. Create system groups that correspond to those activities. For example, create an accounting-ro and accounting-rw group to categorize users within your Accounting department that should have the ability to only read or read and write data. If you're simply managing a home file server, be careful of open network shares where users have read and write access by default. By allowing users to do as little as possible, you'll prevent a great many disasters right away.

In *Chapter 2, Managing Users and Permissions* (as well as *Chapter 21, Securing Your Server*), we talked about best practices for the `sudo` command. While the `sudo` command is useful, it's often misused. By default, anyone that's a member of the `sudo` group can use `sudo` to do whatever they want. We talked about how to restrict `sudo` access to particular commands, which is always recommended. Only trusted administrators should have full access to `sudo`. Everyone else should have `sudo` permissions only if they really need them, and even then, only when it comes to commands that are required for their job. A user with full access to `sudo` can delete an entire filesystem, so it should never be taken lightly.

In regard to network shares, it's always best to default to read-only whenever possible. This isn't just because of the possibility of a user accidentally deleting data; it's always possible for applications to malfunction and delete data as well. With a read-only share, the modification or deletion of files isn't possible. Additional read-write shares can be created for those who need them, but if possible, always default to read-only.

Although I've spent a lot of time discussing security in a software sense, physical security is important too. For the purposes of this book, physical security doesn't really enter the discussion much because our topic is specifically Ubuntu Server, and nothing you install on Ubuntu is going to increase the physical security of your servers. It's worth noting, however, that physical security is every bit as important as securing your operating systems, applications, and data files.

All it would take is someone tripping over a network cable in a server room to disrupt an entire subnet or cause a production application to go offline. Server rooms should be locked, and only trusted administrators should be allowed to access your equipment. I'm sure this goes without saying and may sound obvious, but I've worked at several companies that did not secure their server room. Nothing good ever comes from placing important equipment within arm's reach of unauthorized individuals.

In this section, I've mentioned *Chapter 21, Securing Your Server*, a couple of times. A good majority of a disaster prevention plan includes a focus on security. This includes, but is not limited to, ensuring security updates are installed in a timely fashion, utilizing security applications such as failure monitors and firewalls, and ensuring secure settings for OpenSSH. I won't go over these concepts again here since we've already covered them, but essentially security is a very important part of a disaster prevention plan. After all, users cannot break what they cannot access, and hackers will have a harder time penetrating your network if you designed it in a security-conscious way.

Effective disaster prevention consists of a list of guidelines for things such as user management, server management, application installations, security, and procedure documents. A full walkthrough of proper disaster prevention would be an entire book in and of itself. My goal with this section is to provide you with some ideas you can use to begin developing your own plan. A disaster prevention plan is not something you'll create all at once but is rather something you'll create and refine indefinitely as you learn more about security and what types of things to watch out for.

The configuration files on your server determine the behavior of your services and applications, and backing up can enable you to recover their state. When it comes to managing the state of files, Git is a very powerful tool to do just that, and we'll talk about it next.

Utilizing Git for configuration management

One of the most valuable assets on a server is its configuration. This is second only to the data the server stores. Often, when we implement new technology on a server, we'll spend a great deal of time editing configuration files all over the server to make it work as best as we can. This can include any number of things, from Apache virtual host files to DHCP server configuration, DNS zone files, and more. If a server were to encounter a disaster from which the only recourse was to completely rebuild it, the last thing we'd want to do is re-engineer all of this configuration from scratch. This is where Git comes in.

In a typical development environment, an application being developed by a team of engineers can be managed by Git, each contributing to a repository that hosts the source code for their software. One of the things that makes Git so useful is how you're able to go back to previous versions of a file in an instant, as it keeps a history of all changes made to the files within the repository.

Git isn't just useful for software engineers, though. It's also a really useful tool we can leverage for keeping track of configuration files on our servers. For our use case, we can use it to record changes to configuration files and push them to a central server for backup. When we make configuration changes, we push the changes back to our Git server. If for some reason we need to restore the configuration after a server fails, we can simply download our configuration files from Git back onto our new server. Another useful aspect of this approach is that if an administrator implements a change to a configuration file that breaks a service, we can simply revert to a known working commit and we'll be immediately back up and running. You can even correlate changes in log files to changes made at around the same time in a Git repository, which makes it easier to narrow down the root cause of an issue.

Configuration management on servers is so important, in fact, I highly recommend that every Linux administrator takes advantage of version control for this purpose. Although it may seem a bit tricky at first, it's actually really easy to get going once you practice with it. Once you've implemented Git for keeping track of all your server's configuration files, you'll wonder how you ever lived without it. We covered Git briefly in *Chapter 15, Automating Server Configuration with Ansible*, where I walked you through creating a repository on GitHub to host Ansible configuration. However, using Github is not mandatory in order to benefit from version control. Sure, it's definitely convenient – but not required. The only required component for a server to function as a Git server is the presence of the git package:

```
sudo apt install git
```


And just like that, you now have a Git server. That may have seemed overly simplistic (and it is) but there aren't a great deal of requirements in order to set up Git as a central resource on your network. Since Git uses OpenSSH by default, you'll be able to store repositories on a server that has the `git` package and is accessible to clients on your network.

If for some reason you don't have an extra server to act as your Git server, you can install the `git` package on another instance and add that functionality to an existing server. My personal preference is for each server to focus on one task (and perform that one task well) but sometimes the budget of the organization may not allow for a stand-alone server for every service.

Now, think of a configuration directory that's important to you, and that you want to place into version control. A good example is the `/etc/apache2` directory on a web server. That's what I'll use in my examples in this section. But you're certainly not limited to that. Any configuration directory you would rather not lose is a good candidate. If you choose to use a different configuration path, change the paths I give you in my examples to that path.

On the server, create a directory to host your repositories. I'll use `/git` in my examples:

```
sudo mkdir /git
```

Next, you'll want to modify this directory to be owned by the administrative user you use on your Ubuntu servers. Typically, this is the user that was created during the installation of the distribution. You can use any user you want actually, just make sure the user is allowed to use OpenSSH to access your Git server. Change the ownership of the `/git` directory so it is owned by this user. My user on my Git server is `jay`, so in my case, I would change the ownership with the following command:

```
sudo chown jay:jay /git
```

Next, we'll create our Git repository within the `/git` directory. For Apache, I'll create a bare repository for it within the `/git` directory. A bare repository is basically a skeleton of a Git repository that doesn't contain any useful data, just some default configuration to allow it to act as a Git folder. To create the bare repository, `cd` into the `/git` directory and execute:

```
git init --bare apache2
```

You should see the following output:

```
Initialized empty Git repository in /git/apache2/
```

That's all we need to do on the server for now for the purposes of our Apache repository. On your client (the server that houses the configuration you want to place under version control), we'll copy this bare repository by cloning it. To set that up, create a `/git` directory on your Apache server (or whatever kind of server you're backing up) just as we did before. Then, `cd` into that directory and clone your repository with the following command:

```
git clone 192.168.1.101:/git/apache2
```

For that command, replace the IP address with either the IP address of your Git server or its hostname if you've created a DNS entry for it. You should see the following output, warning us that we've cloned an empty repository:

```
warning: You appear to have cloned an empty repository
```

This is fine, we haven't actually added anything to our repository yet. If you were to `cd` into the directory we just cloned and list its storage, you'd see it as an empty directory. If you use `ls -a` to view hidden directories as well, you'll see a `.git` directory inside. Inside the `.git` directory, we'll have configuration items for Git that allow this repository to function properly. For example, the config file in the `.git` directory contains information on where the remote server is located. We won't be manipulating this directory; I just wanted to give you a quick overview of what its purpose is.

Note that if you delete the `.git` directory in your cloned repository, that basically removes version control from the directory and makes it a normal directory.

Anyway, let's continue. We should first make a backup of our current `/etc/apache2` directory on our web server, in case we make a mistake while converting it to being version controlled:

```
sudo cp -rp /etc/apache2 /etc/apache2.bak
```

Then, we can move all the contents of `/etc/apache2` into our repository:

```
sudo mv /etc/apache2/* /git/apache2
```

The `/etc/apache2` directory is now empty. Be careful not to restart Apache at this point; it won't see its configuration files and will fail. Remove the (now empty) `/etc/apache2` directory:

```
sudo rm /etc/apache2
```

Now, let's make sure that Apache's files are owned by root. The problem though is if we use the `chown` command, as we normally would to change ownership, we'll also change the `.git` directory to be owned by root as well. We don't want that, because the user responsible for pushing changes should be the owner of the `.git` folder. The following command will change the ownership of the files to root, but won't touch hidden directories such as `.git`:

```
sudo find /git/apache2 -name '.*' -prune -o -exec chown root:root {} +
```

When you list the contents of your repository directory now, you should see that all files are owned by root, except for the `.git` directory, which should be owned by your administrative user account.

Next, create a symbolic link to your Git repository so the `apache2` daemon can find it:

```
sudo ln -s /git/apache2 /etc/apache2
```

At this point, you should see a symbolic link for Apache, located at `/etc/apache2`. If you list the contents of `/etc` while grepping for `apache2`, you should see it as a symbolic link:

```
ls -l /etc | grep apache2
```

The directory listing will look similar to the following:

```
lrwxrwxrwx 1 root root 37 2020-06-25 20:59 apache2 -> /git/apache2
```

If you reload Apache, nothing should change and it should find the same configuration files as it did before, since its directory in `/etc` maps to `/git/apache2`, which includes the same files it did before:

```
sudo systemctl reload apache2
```

If you see no errors, you should be all set. Otherwise, make sure you created the symbolic link properly.

Next, we get to the main attraction. We've copied Apache's files into our repository, but we didn't actually push those changes back to our Git server yet. To set that up, we'll need to associate the files within our `/git/apache2` directory into version control. The reason for this is the files simply being in the `git` repository folder isn't enough for Git to care about them. We have to tell Git to pay attention to individual files. We can add every file within our Git repository for Apache by entering the following command from within that directory:

```
git add .
```

This basically tells Git to add everything in the directory to version control. You can actually do the following to add an individual file:

```
git add <filename>
```

In this case, we want to add everything, so we used a period in place of a directory name to add the entire current directory.

If you run the `git status` command from within your Git repository, you should see output indicating that Git has new files that haven't been committed yet. A **Git commit** simply finalizes the changes locally. Basically, it packages up your current changes to prepare them for being copied to the server. To create a commit of all the files we've added so far, `cd` into your `/git/apache2` directory and run the following to stage a new commit:

```
git commit -a -m "My first commit."
```

With this command, the `-a` option tells Git that you want to include anything that's changed in your repository. The `-m` option allows you to attach a message to the commit, which is actually required. If you don't use the `-m` option, it will open your default text editor and allow you to add a comment from there.

Finally, we can push our changes back to the Git server:

```
git push origin master
```

By default, the `git` suite of commands utilizes OpenSSH, so our `git push` command should create an SSH connection back to our server and push the files there. You won't be able to inspect the contents of the Git directory on your Git server, because it won't contain the same file structure as your original directory. Whenever you pull a Git repository though, the resulting directory structure will be just as you left it.

From this point forward, if you need to restore a repository onto another server, all you should need to do is perform a Git clone. To clone the repository into your current working directory, execute the following:

```
git clone 192.168.1.101:/git/apache2
```

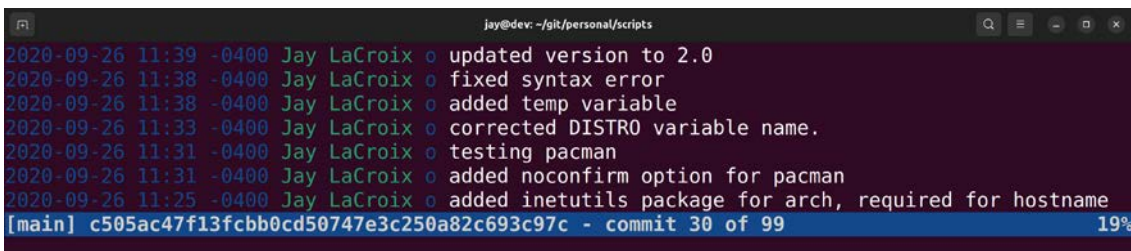
Now, each time you make changes to your configuration files, you can perform a `git commit` and then push the changes up to the server to keep the content safe:

```
git commit -a -m "Updated config files."git push origin master
```

Now we know how to create a repository, push changes to a server, and pull the changes back down. Finally, we'll need to know how to revert changes should our configuration get changed with non-working files. First, we'll need to locate a known working commit. My favorite method is using the `tig` command. The `tig` package must be installed for this to work, but it's a great utility to have:

```
sudo apt install tig
```

The `tig` command (which is just `git backward`) gives us a semi-graphical interface to browse through our Git commits. To use it, simply execute the `tig` command from within a Git repository. In the following example screenshot, I've executed `tig` from within a Git repository on one of my servers:



```

jay@dev: ~/git/personal/scripts
2020-09-26 11:39 -0400 Jay LaCroix o updated version to 2.0
2020-09-26 11:38 -0400 Jay LaCroix o fixed syntax error
2020-09-26 11:38 -0400 Jay LaCroix o added temp variable
2020-09-26 11:33 -0400 Jay LaCroix o corrected DISTRO variable name.
2020-09-26 11:31 -0400 Jay LaCroix o testing pacman
2020-09-26 11:31 -0400 Jay LaCroix o added noconfirm option for pacman
2020-09-26 11:25 -0400 Jay LaCroix o added inetutils package for arch, required for hostname
[main] c505ac47f13fcbb0cd50747e3c250a82c693c97c - commit 30 of 99 19%
```

Figure 23.1: An example of the `tig` command, looking at a repository for example scripts

While using `tig`, you'll see a list of Git commits, along with their dates and comments that were entered with each. To inspect one, press the *up* and *down* arrows to change your selection, then press *Enter* on the one you want to view. You'll see a new window, which will show you the commit hash (which is a long string of alphanumeric characters), as well as an overview of which lines were added or removed from the files within the commit. To revert one, you'll first need to find the commit you want to revert to and get its commit hash. The `tig` command is great for finding this information. In most cases, the commit you'll want to revert to is the one *before* the change took place. In my example screenshot, I fixed the syntax issue on 9/26/2020. If I want to restore that file, I should revert to the commit below that. I can get the commit hash by highlighting that entry and pressing *Enter*. It's at the top of the window. Then, I can exit `tig` by pressing *q*, and then revert to that commit:

```
git checkout <commit hash>
```

And just like that, the entire directory tree for the repository instantly changes to exactly what it was before the bad commit took place. I can then restart or reload the daemon for this repository, and it will be back to normal.

At this point, you'd want to test the application to make sure that the issue is completely fixed. After some time has passed and you're finished testing, you can make the change permanent. First, we switch back to the most recent commit:

```
git checkout main
```

Then, we permanently switch back to the commit that was found to be working properly:

```
git revert --no-commit <commit hash>
```

Then, we can commit our reverted Git repository and push it back to the server:

```
git commit -a -m "The previous commit broke the application. Reverting."git  
push origin master
```

As you can see, Git is a very useful ally to utilize when managing configuration files on your servers. This benefits disaster recovery, because if a bad change is made that breaks a daemon, you can easily revert the change. If the server were to fail, you can recreate your configuration almost instantly by just cloning the repository again. There's certainly a lot more to Git than what we've gone over in this section, so feel free to pick up a book about it if you wish to take your knowledge to the next level. But in regard to managing your configuration with Git, all you'll need to know is how to place files into version control, update them, and clone them to new servers. Some services you run on a server may not be a good candidate for Git, however. For example, managing an entire MariaDB database via Git would be a nightmare, since there is too much overhead with such a use case, and database entries would likely change too rapidly for Git to keep up. Use your best judgment. If you have some configuration files that are only manipulated every once in a while, they'll be a perfect candidate for Git.

Backups are one of those things that some people don't seem to take seriously until it's too late. Data loss can be a catastrophic event for an organization, so it's imperative that you implement a solid backup plan. In the next section, we'll look at what that entails.

Implementing a backup plan

Creating a solid backup plan is one of the most important things you'll ever do as a server administrator. Even if you're only using Ubuntu Server at home as a personal file server, backups are critical. During my career, I've seen disks fail many times. I often hear arguments about which hard disk manufacturer beats others in terms of longevity, but I've seen disk failures so often, I don't trust any of them. All disks will fail eventually, it's just a matter of when. And when they do fail, they'll usually fail hard with no easy way to recover data from them. A sound approach to managing data is that any disk or server can fail, and it won't matter, since you'll be able to regenerate your data from other sources, such as a backup or secondary server.

There's no one best backup solution, since it all depends on what kind of data you need to secure, and what software and hardware resources are available to you. For example, if you manage a database that's critical to your company, you should back it up regularly. If you have another server available, set up a replication secondary server so that your primary database isn't a single point of failure. Not everyone has an extra server lying around, so sometimes you have to work with what you have available.

This may mean that you'll need to make some compromises, such as creating regular snapshots of your database server's storage volume or regularly dumping a backup of your important databases to an external storage device.

The `rsync` utility is one of the most valuable pieces of software around to server administrators. It allows us to do some really wonderful things. In some cases, it can save us quite a bit of money. For example, online backup solutions are wonderful in the sense that we can use them to store off-site copies of our important files. However, depending on the volume of data, they can be quite expensive. With `rsync`, we can back up our data in much the same way, with not only our current files copied over to a backup target but also differentials. If we have another server to send the backup to, even better.

At one company I managed servers for, they didn't want to subscribe to an online backup solution. To work around that, a server was set up as a backup point for `rsync`. We set up `rsync` to back up to the secondary server, which housed quite a lot of files. Once the initial backup was complete, the secondary server was sent to one of our other offices in another state. From that point forward, we only needed to run `rsync` weekly, to back up everything that had been changed since the last backup. Sending files via `rsync` to the other site over the internet was rather slow, but since the initial backup was already complete before we sent the server there, all we needed to back up each week was differentials. Not only is this an example of how awesome `rsync` is and how we can configure it to do pretty much what paid solutions do but the experience was also a good example of utilizing what you have available to you.

Since we've already gone over `rsync` in *Chapter 12, Sharing and Transferring Files*, I won't repeat too much of that information here. But since we're on the subject of backing up, the `--backup-dir` option is worth mentioning again. This option allows you to copy files that would normally be replaced to another location. As an example, here's the `rsync` command I mentioned in *Chapter 12, Sharing and Transferring Files*:

```
CURDATE=$(date +%m-%d-%Y)
export $CURDATE
sudo rsync -avb --delete --backup-dir=/backup/incremental/$CURDATE /src /target
```

This command was part of the topic of creating an `rsync` backup script. The first command simply captures today's date and stores it in a variable named `$CURDATE`. In the actual `rsync` command, we refer to this variable. The `-b` option (part of the `-avb` option string) tells `rsync` to make a copy of any file that would normally be replaced. If `rsync` is going to replace a file on the target with a new version, it will move the original file to a new name before overwriting it. The `--backup-dir` option tells `rsync` that when it's about to overwrite a file, to put it somewhere else instead of copying it to a new name. We give the `--backup-dir` option a path, where we want the files that would normally be replaced to be copied. In this case, the backup directory includes the `$CURDATE` variable, which will be different every day. For example, a backup run on 8/16/2022 would have a backup directory of the following path, if we used the command I gave as an example:

```
/backup/incremental/8-16-2022
```

This essentially allows you to keep differentials. Files on `/src` will still be copied to `/target`, but the directory you identify as `--backup-dir` will contain the original files before they were replaced that day.

On my servers, I use the `--backup-dir` option with `rsync` quite often. I'll typically set up an external backup drive, with the following three folders:

- `current`
- `archive`
- `logs`

The `current` directory always contains a current snapshot of the files on my server. The `archive` directory on my backup disks is where I point the `--backup-dir` option. Within that directory will be folders named with the dates that the backups were taken. The `logs` directory contains log files from the backup. Basically, I redirect the output of my `rsync` command to a log file within that directory, each log file being named with the same `$(CURDATE)` variable so I'll also have a backup log for each day the backup runs. I can easily look at any of the logs for which files were modified during that backup, and then traverse the `archive` folder to find an original copy of a file. I've found this approach to work very well. Of course, this backup is performed with multiple backup disks that are rotated every week, with one always off-site. It's always crucial to keep a backup off-site in case of a situation that could compromise your entire local site.

The `rsync` utility is just one of many you can utilize to create your own backup scheme. The plan you come up with will largely depend on what kind of data you're wanting to protect and what kind of downtime you're willing to endure.

Ideally, we would have an entire warm site with servers that are carbon copies of our production servers, ready to be put into production should any issues arise, but that's also very expensive, and whether you can implement such a routine will depend on your budget. However, Ubuntu has many great utilities available that you can use to come up with your own system that works. If nothing else, utilize the power of `rsync` to back up to external disks and/or external sites.

A tool that is very valuable when working to recover physical servers is USB recovery media, such as flash drives with a bootable ISO image written to them. In the next section, we'll take a look.

Utilizing bootable recovery media

The concept of **live media** is a wonderful thing, as we can boot into a completely different working environment from the operating system installed on our device and perform tasks without disrupting installed software on the host system. The desktop version of Ubuntu, for example, offers a complete computing environment we can use in order to not only test hardware and troubleshoot our systems but also browse the web just as we would on an installed system. In terms of recovering from disasters, live media becomes a saving grace.

As administrators, we run into one problem after another. This gives us our job security. Computers often seemingly have a mind of their own, failing when least expected (as well as seemingly every holiday). Our servers and desktops can encounter a fault at any time, and live media allows us to separate hardware issues from software issues, by troubleshooting from a known good working environment.

One of my favorites when it comes to live media is the desktop version of Ubuntu. Although geared primarily toward end users who wish to install Ubuntu on a laptop or desktop, as administrators we can use it to boot a machine that normally wouldn't, or even recover data from failed disks. For example, I've used the Ubuntu live media to recover data from both failed Windows and Linux systems, by booting the machine with the live media and utilizing a network connection to move data from the bad machine to a network share. Often, when a computer or server fails to boot, the data on its disk is still accessible. Assuming the disk wasn't encrypted during installation, you should have no problem accessing data on a server or workstation using live media such as Ubuntu live media.

Sometimes, certain levels of failure require us to use different tools. While Ubuntu's live media is great, it doesn't work for absolutely everything. One situation is a failing disk. Often, you'll be able to recover data using Ubuntu's live media from a failing disk, but if it's too far gone, then the Ubuntu media will have difficulty accessing data from it as well. Tools such as Clonezilla specialize in working with hard disks and may be a better choice.

Live media can totally save the day. The Ubuntu live image in particular is a great boot disk to have available to you, as it gives you a very extensive environment you can use to troubleshoot systems and recover data.

One of the best aspects of using the Ubuntu live image is that you won't have to deal with the underlying operating system and software set at all, you can bypass both by booting into a known working desktop, and then copy any important files from the drive right onto a network share. Another important feature of Ubuntu live media is the memory test option. Quite often, strange failures on a computer can be traced to defective memory. Other than simply letting you install Ubuntu, the live media is a Swiss Army knife of many tools you can use to recover a system from disaster. If nothing else, you can use live media to pinpoint whether a problem is software- or hardware-related. If a problem can only be reproduced in the installed environment but not in a live session, chances are a configuration problem is to blame. If a system also misbehaves in a live environment, it may help you identify a hardware issue. Either way, every good administrator should have live media available to troubleshoot systems and recover data when the need arises.

Summary

In this chapter, we looked at several ways in which we can prevent and recover from disasters. Having a sound prevention and recovery plan in place is an important key to managing servers efficiently. We need to ensure we have backups of our most important data ready for whenever servers fail, and we should also keep backups of our most important configurations. Ideally, we'll always have a warm site set up with preconfigured servers ready to go in a situation where our primary servers fail, but one of the benefits of open source software is that we have a plethora of tools available to us that we can use to create a sound recovery plan. In this chapter, we looked at leveraging `rsync` as a useful utility for creating differential backups, and we also looked into setting up a Git server we can use for configuration management, which is also a crucial aspect of any sound prevention plan. We also talked about the importance of live media in diagnosing issues.

And with this chapter, this book comes to a close. Writing this book has been an extremely joyful experience. I was thrilled to write the first edition when Ubuntu 16.04 was in development, it was a fun project to write the second edition and update it to cover Ubuntu 18.04, the third edition for 20.04, and I'm even more thrilled to have had the opportunity to update this body of work yet again for 22.04 in this latest edition. I'd like to thank each and every one of you, my readers, for taking the time to read this book. In addition, I would like to thank all of the viewers of my YouTube channel, *Learn Linux TV* (<https://www.learnlinux.tv>), because I probably wouldn't have had the opportunity to write this in the first place had it not been for my viewers helping make my channel so popular.

I'd also like to thank Packt Publishing for giving me the opportunity to write a book about one of my favorite technologies. Writing this book was definitely an honor. When I first started with Linux in 2002, I never thought I'd actually be an author, teaching the next generation of Linux administrators the tricks of the trade. I wish each of you the best of luck, and I hope this book is beneficial to you and your career.

For additional content, be sure to check out <https://learnlinux.tv> for even more content. I have quite a few training videos available there, free of charge.

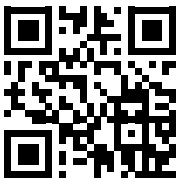
Further reading

- *Pro Git* by Scott Chacon and Ben Straub: <https://learnlinux.link/git-book>
- *Introduction to RAID terminology and concepts*: <https://learnlinux.link/raid-concepts>

Join our community on Discord

Join our community's Discord space for discussions with the author and other readers:

<https://packt.link/LWaZ0>





packt.com

Subscribe to our online digital library for full access to over 7,000 books and videos, as well as industry leading tools to help you plan your personal development and advance your career. For more information, please visit our website.

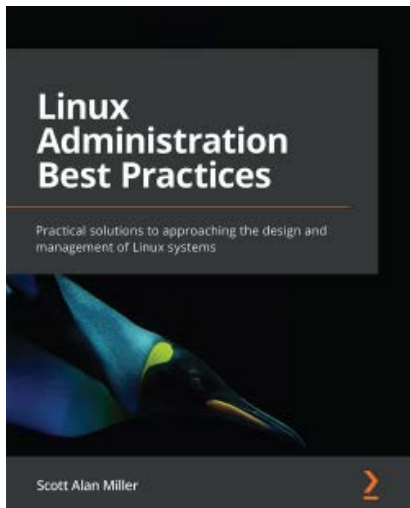
Why subscribe?

- Spend less time learning and more time coding with practical eBooks and Videos from over 4,000 industry professionals
- Improve your learning with Skill Plans built especially for you
- Get a free eBook or video every month
- Fully searchable for easy access to vital information
- Copy and paste, print, and bookmark content

At www.packt.com, you can also read a collection of free technical articles, sign up for a range of free newsletters, and receive exclusive discounts and offers on Packt books and eBooks.

Other Books You May Enjoy

If you enjoyed this book, you may be interested in these other books by Packt:

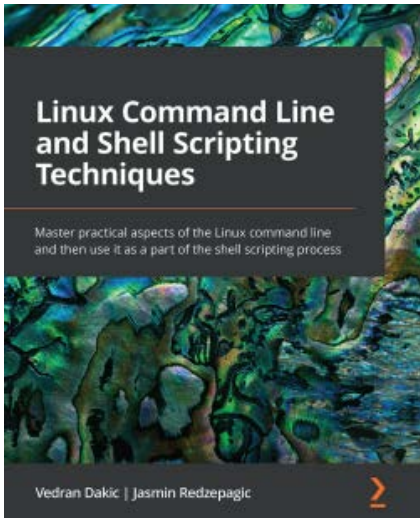


Linux Administration Best Practices

Scott Alan Miller

ISBN: 9781800568792

- Find out how to conceptualize the system administrator role
- Understand the key values of risk assessment in administration
- Apply technical skills to the IT business context
- Discover best practices for working with Linux specific system technologies
- Understand the reasoning behind system administration best practices
- Develop out-of-the-box thinking for everything from reboots to backups to triage
- Prioritize, triage, and plan for disasters and recoveries
- Discover the psychology behind administration duties



Linux Command Line and Shell Scripting Techniques

Vedran Dakic

Jasmin Redzepagic

ISBN: 9781800205192

- Get an introduction to the command line, text editors, and shell scripting
- Focus on regular expressions, file handling, and automating complex tasks
- Automate common administrative tasks
- Become well-versed with networking and system security scripting
- Get to grips with repository management and network-based file synchronization
- Use loops, arguments, functions, and arrays for task automation

Packt is searching for authors like you

If you're interested in becoming an author for Packt, please visit authors.packtpub.com and apply today. We have worked with thousands of developers and tech professionals, just like you, to help them share their insight with the global tech community. You can make a general application, apply for a specific hot topic that we are recruiting an author for, or submit your own idea.

Share Your Thoughts

Now you've finished *Mastering Ubuntu Server, Fourth Edition*, we'd love to hear your thoughts! If you purchased the book from Amazon, please [click here](#) to go straight to the Amazon review page for this book and share your feedback or leave a review on the site that you purchased it from.

Your review is important to us and the tech community and will help us make sure we're delivering excellent quality content.

Index

Symbols

24-bit subnet 211

/etc/exports file 258

/etc/fstab file 184

adding 186-188

contents, analyzing 184-186

/etc/passwd file 39, 40

/etc/shadow file 41-43

/etc/skel

default configuration files, distributing
with 43, 44

A

access log 289

Active Directory (AD) 32

additional storage volumes

adding 173-176

adduser command

using 36, 37

administrator access

configuring, with sudo 53-55

Advanced Package Tool (APT) 68

Amazon Lightsail 410

Amazon Machine Image (AMI) 444, 447

Amazon Web Services (AWS) 410, 461

certification 458

concepts 413-416

documentation 458

EC2 instances, running only when needed 456

experimenting and learning 458

online training and labs 457

Ubuntu Server instance, creating 432-443

unnneeded EC2 instances, stopping or
terminating 457

Ansible

combining, with Terraform for full deployment
solution 481-483

need for 315

settings, configuring 321-323

working with 319-321

Ansible's pull method

using 330-334

Apache

additional modules, installing 291-293

configuring 285-291

installing 285-291

securing, with TLS 293-299

application log files

viewing 94, 95

Application Logs 516

apt

Debian packages, managing with 68-71

A Record 290

arguments 88

configuring, to ps command 137-141

attack surface 486

lowering 485-489

Auto Scaling

used, for scaling Ubuntu EC2 deployments
automatically 446, 447

group, creating 448-454

AWS account

creating 416, 417

signing up 417-421

user security, implementing 421-427

AWS Billing Dashboard

- billing alert, adding 456
- billing information, viewing 455
- unnneeded backups, removing 456

AWS Certified Cloud Practitioner 458**AWS Certified Sysops Administrator 458****AWS EC2 instance**

- Ubuntu, deploying as 428

AWS Management Console 417**B****backported 81****backup plan**

- implementing 538-540

Bash 115

- history 117-119

Berkeley Internet Name Daemon (BIND) package

- external DNS, setting up with 237, 238

bootable flash drive

- creating 8-11
- partitioning layout, planning 11, 12

bootable recovery media

- utilizing 540, 541

Bourne Again Shell 115**broadcast address 230****C****Caching Name Server 237****Canonical Livepatch service**

- reference link 494
- used, for automatically installing patches 494

Canonical Name (CNAME) record 242**Certificate Signing Request (CSR) 297****Classless Inter-Domain**

- Routing (CIDR notation) 229

client servers

- configuring 323-327

cloning

- used, for simplifying virtual machine creation 354-356

cloud computing

- using, considerations 411-413

Cloud Deployment

- automating 462

CloudFormation 463**CloudFront 428****cloud infrastructure**

- versus on-premises infrastructure 410

command line

- used, for managing Virtual Machine (VM) 356, 357

command-line tricks

- utilizing 119-122

commands

- issuing, with OpenSSH 217-219

Common Vulnerabilities and Exposures (CVEs) 490

- responding to 490

config file

- used, for simplifying SSH connections 223-225

configuration management

- Git, utilizing 533-538
- need for 313, 315

container 360

- deploying, via Kubernetes cluster 402-408
- using 360

containerd 394**containerization 359, 360****container orchestration 380, 381****Container Runtime 394****Content Delivery Network (CDN) 428****contents**

- viewing, of files 91-94

control groups (cgroups) 361

cron

used, for scheduling tasks 151-153

cron job 330

cross-platform 463

curdate 130

D

daemons 148

database server

preparations, for setting up 265-267

Debian 64

Debian packages 65, 66

backing up 78, 79

managing, with apt 68-71

restoring 78, 79

versus Snap packages 65

default configuration files

distributing, with /etc/skel 43, 44

defective RAM

diagnosing 527-529

device, selecting for server

laptop 4

physical desktop 4

physical servers 3

Raspberry Pi 5

Virtual Machine (VM) 5

Virtual Private Server (VPS) 5

df command

using 156-158

DHCP reservation 209

DigitalOcean 410

directories

copying 97-99

moving 97-99

ownership, modifying 60, 61

permissions, setting on 56

renaming 97-99

Disaster Recovery 462

disasters

preventing 531-533

Disk Operating System (DOS) 178

disks

decrypting, with Linux Unified Key Setup (LUKS) 507, 508

encrypting, with Linux Unified Key Setup (LUKS) 507, 508

disk usage

df command, using 156-158

investigating 158-161

viewing 155

DNS server

adding 236, 237

external DNS, setting up with bind 237, 238

hosts, adding 239-243

internal DNS, setting up 239-243

Docker

installing 362, 363

versus LXD 361, 362

Docker containers

managing 363-370

Dockerfiles

Docker image creation,
automating with 371, 372

Docker Hub 362

reference link 363

Docker image creation

automating, with Dockerfiles 371, 372

document root 286

Domain Name System (DNS) 213

Dynamic Host Control Protocol (DHCP) 15

Dynamic Host Control

Protocol (DHCP) server 228

setting up, for IP addresses 231-236

E

EC2 instance deployment

automating 470-476

EC2 instances

running, only when needed 456

stopping or terminating 457

Elastic Block Store (EBS) 415

Elastic Compute Cloud (EC2) 415

Elastic Container Service (ECS) 362

Elastic Kubernetes Service (EKS) 416

Elastic Load Balancer (ELB) 415

Error Correction Code (ECC) 4

error log 289

Etcher 8

export 255

export root 255

F

Fail2ban 498

configuring 498-502

installing 498-502

files

contents, viewing 91-94

copying 97-99

editing, with Nano text editors 99, 100

editing, with Vim text editors 99, 100

moving 97-99

ownership, modifying 60, 61

permissions, setting on 56

renaming 97-99

sharing, with Windows

users via Samba 249-254

transferring, with rsync 258-261

transferring, with Secure Copy (SCP) 261-263

file server considerations 247, 248

filesystem cache 162

Filesystem Hierarchy Standard (FHS) 90, 182

firewall 505

setting up 505, 506

Fish 116

Flannel 400

for loop 128

G

gateway 243

Git

utilizing, for configuration
management 533-538

Git commit 536

Git repository

creating 316-319

Google Cloud Platform (GCP) 410

grants 273

Graphical User Interface (GUI) 100, 139, 341

groups

managing 46-49

purpose 32

GUID Partition Table (GPT) 178

H

hard links

using 111-113

hardware enablement (HWE) 81

updates, advantage 81-83

hash bang 125

Homebrew

installation link 384

utilizing 384

hostname

setting 201-204

htop

used, for viewing resource usage 168-171

I**IAM role**

setting up, for Session Manager 428-432

Identity and Access Management (IAM) 415**if statement 126****infinite loop 128****Infrastructure as Code (IaC) 314, 463****inodes 111****Input/Output (IO) 526****input streams 108-110****installation media**

obtaining 6-8

Integrated Development

Environment (IDE) 104

internet gateway

setting up 243, 244

Internet Service Provider (ISP) 237**inventory file 315**

setting up 321-323

IP addresses

DHCP server, setting up for 231-236

scheme, planning 227-231

J**jobs**

managing 133-136

K**Kernel-based Virtual Machine (KVM) 338, 362****Keyboard, Video, and Mouse (KVM) 4****ksh 116****kubeadm package 397****Kube Control (kubectrl) 392****kubectrl package 397****kubelet package 398****Kubernetes cluster**

containers, deploying via 402-408

installing 397-402

packages 397

preliminary setup 393-397

setting up 392, 393

Kubernetes (K8s) 381

testing, by preparing lab environment 381-383

L**lab environment**

preparing, for Kubernetes testing 381-383

laptop 4**launch template**

creating 447, 448

options 447

LearnLinuxTV 213**Lightweight Directory Access
Protocol (LDAP) 32****Linode 410****Linux**

MicroK8s, installing on 383, 384

name resolution 213-215

Linux commands 85-89**Linux Containers (LXC) 361****Linux filesystem layout 89-91****Linux package management 63-65****Linux shell 115, 116****Linux signals 146****Linux Unified Key Setup (LUKS)**

used, for decrypting disks 507

used, for encrypting disks 507

live media

concept 540

load average 166-168**Logical Volume Manager (LVM) 173, 267**

benefits 174

logical volumes, formatting 195-197
utilizing 190, 191
volumes, removing 197
working with 191-195

logical volumes

formatting 195-197

long-term support (LTS) 7, 81

loopback addresses 203

LVM snapshots 198, 199

LXD 361

versus Docker 361

LXD containers

managing 373-377

M

macOS

MicroK8s, installing on 384-386

MariaDB

best practices, for secure
database servers 502-504
configuration files 269-271
databases, managing 272-278
installing 267-269

Master Boot Record (MBR) 178

memory

usage, monitoring 161

Memtest86+ 528, 529

MicroK8s

installing, on Linux 383, 384
installing, on macOS 384-386
installing, on Windows 386-389
interacting with 390-392
utilizing 383

Microsoft Azure 410

misbehaving processes

dealing with 145-147

Multi-Factor Authentication (MFA) 422

myvar 128

N

Nano text editors

editing with 100-102
used, for editing files 99, 100

NCurses Disk Usage 159

network

interfaces, managing 204-208
issues, tracing 520-524
services, setting up 227

Network Time Protocol (NTP) 133

Nextcloud

configuring 303-310
setting up 303-310

NFS 247

NFS shares

setting up 254-258

NGINX

configuring 299-303
installing 299-303

NodePort Service 406

non-Graphical User Interface (non-GUI) 67

Non-Volatile Memory Express (NVMe) 176

O

object storage 415

on-premises infrastructure

versus cloud infrastructure 410

OpenSSH 495

commands, issuing with 217-219
installing 216, 217
securing 495-498
working with 215

OpsWorks 464

orphaned apt packages

cleaning up 79-81

Out of Memory (OOM) Killer 163

output streams 108-111

output variable 478

P

package maintainer 64

packages

additional repositories, adding 75-77

repositories, managing 75

searching 73-75

partitioning 11

layout, planning 11, 12

tips 12

partitions

creating 177-180

formatting 180-182

passphrase, of OpenSSH key

modifying 223

passwords

expiration information, setting 50, 51

managing 49

policies 49

policy, setting 51, 52

user accounts, locking 49, 50

user accounts, unlocking 49, 50

patches

installing, automatically with Canonical Livepatch service 494

Perl 37

permissions

modifying 59, 60

ownership of objects, modifying 60, 61

setting, on files and directories 56

viewing 56-58

Personal Package Archive (PPA)

adding 77, 78

physical desktop 4

physical servers 3

playbook 323

Pluggable Authentication Module (PAM) 51

Pod Network 398

principle of least privilege 489, 532

print working directory 86

private key

generating 220

privileged commands

running, with sudo 33, 34

processes, on Linux system

priority, modifying 141-145

Process ID (PID) 137

PS1 prompt 202

ps command 136

arguments, configuring to 137-141

running processes, viewing with 136, 137

public key

copying, to remote server 221, 222

generating 220

pull method 315

Q

Quick Emulator (QEMU) 338

R

Raspberry Pi 5

Ubuntu, installing on 25-28

Red Hat 64

Redundant Array of Independent
Disks (RAID) 176

region

selecting 427, 428

resources

issues, troubleshooting 524-527

Return on Investment (ROI) 411

root cause analysis

conducting 513-515

Route 53 415

RPM packages 65**rsync**

- used, for transferring files 258-261
- backup script, writing 129-131

running processes

- viewing, with ps command 136, 137

S**Samba** 247

- files sharing, with Windows users via 249-254

scope

- evaluating 511-513

scripts

- writing 124-129

secondary database server

- setting up 278-282

Secure Copy (SCP)

- used, for transferring files 261-263

secure database servers

- MariaDB, best practices 502-504

Secure Sockets Layer (SSL) 293**security groups** 416

- managing, with Terraform 476-479

security updates

- installing 490-494

Serial Advanced Technology**Attachment (SATA)** 184**server**

- device, selecting 3
- role, determining 2, 3

server memory 161-163**Server Message Block (SMB)** 248**Services for NFS** 248**Session Manager**

- IAM role, setting up 428-432

shebang 125**shell script** 37**Simple Storage Service (S3)** 415**Snap packages** 66, 67

- managing, with snap 71-73
- versus Debian packages 65

software

- installing 68
- removing 68

SSH agent

- utilizing 222

SSH connections

- simplifying, with config file 223-225

SSH key management

- OpenSSH key passphrase, changing 223
- private key, generating 220
- public key, copying to remote server 221, 222
- public key, generating 220, 221
- SSH agent, utilizing 222
- working with 219

SSH passphrase

- modifying 223

Stable Release Updates (SRUs) 64**standard error (stderr)** 124**standard input (stdin)** 108, 124**standard output (stdout)** 108, 124**Start of Authority (SOA) line** 240**static IP addresses**

- assigning 209-213

storage devices

- formatting 176, 177
- partitioning 176, 177

storage volume

- backing up 188-190
- mounting 182, 183
- restoring 188-190
- unmounting 182, 183

streams 108**subnets** 229**sub-shell** 130

subtree checking 255**sudo**

- administrator access, configuring with 53-55
- locking down 509
- using, to run privileged commands 33, 34

swap

- managing 163-166

swap file 12**swap partition 12****symbolic**

- using 111-113

symbolic link (symlink) 112**system logs**

- viewing 515-520

system processes

- managing 148-151

T**Tab Completion 119****teletypewriter 138****Terraform 461, 463**

- installing 465-470
- used, for combining Ansible with full deployment solution 481-483
- used, for managing security groups 476-479
- workflow 463, 464

Terraform destroy 479

- using 479-481

Terraform plan 474**Time to Live (TTL) 240****Transport Layer Security (TLS) 285**

- used, for securing Apache 293-299

U**Ubuntu**

- deploying, as AWS EC2 instance 428
- installing, on Raspberry Pi 25-28

Ubuntu AMIs

- creating 444-446
- deploying 444-446

Ubuntu CVE tracker

- reference link 490

Ubuntu EC2 deployments

- scaling, automatically with Auto Scaling 446, 447

Ubuntu One account

- URL 494

Ubuntu Server

- installing 12-24
- instance, creating in AWS 432-443

ufw command

- reference link 506

Uncomplicated Firewall (UFW) 505**Uninterruptible Power Supply (UPS) 4****units 148****Universally Unique Identifier (UUID) 184****Universal Naming Convention (UNC) 253****universal packages 66****unix_socket authentication 268****useradd command**

- using 34, 35

use root

- usage 32, 33

users

- creating 34
- purpose 32
- removing 34-39
- switching 45, 46

V**variables 122-124****Vim**

- editing with 102-108

Vim text editors **Zsh 116**

used, for editing files 99, 100

VirtualBox 5

download link 5

virtual host 286**virtualization 359**

consideration 338, 339

prerequisites 337-339

virtual machine creation

simplifying, with cloning 354-356

Virtual Machine Manager 341**virtual machine network**

bridging 351-353

virtual machine server

setting up 339-345

Virtual Machine (VM) 5, 319, 514, 337

creating 345-350

managing, via command line 356, 357

weaknesses 359

Virtual MFA device 423**Virtual Private Cloud (VPC) 415****Virtual Private Server (VPS) 5, 45, 267, 290, 410****VM Template 354****Voice over IP (VoIP) 228****vulnerability chain 487**

W

web server deployment

automating 327-330

Windows

MicroK8s, installing on 386-389

Windows users

files sharing, via Samba 249-254

Z

zombie process 146**zone file 237, 239**

