



Getting Started On Windows

Welcome to ACME! This guide will help you get set up for ACME.

NOTE

One little-known resource BYU students should be aware of is BYU IT Surplus. They offer “rental” laptops, desktops, and monitors to BYU students **for free**. However, hardware is distributed on a first-come, first-served basis. To find equipment, simply stop by their location during their hours with your BYU ID or give them a call (find their information at <https://itsurplus.byu.edu> or internet search “BYU IT Surplus”).

Most of the code we run in ACME isn’t terribly demanding, but occasionally older computers may struggle. In addition to BYU IT Surplus, this document will also show you how to remotely run your code on ACME lab computers to take advantage of their newer hardware, and in future labs we’ll show you how to use online services such as Google Colab and Kaggle.

Installing or Updating WSL

Windows Subsystem for Linux (WSL) is a compatibility layer for running Linux natively on Windows 10 and 11. Specifically, we’ll install Ubuntu on Windows using WSL, and this is how you will run all your code in ACME. You may have already developed your own workflow on Windows that you’re familiar with, but we highly recommend that you use WSL, as not doing so will most certainly give you extensive grief when trying to install different items for many ACME labs.

Don’t worry if you have zero Linux experience. Even with very little or no Linux experience, you will soon become comfortable enough to help your peers and coworkers with it in the future.

1. *Installing WSL and Ubuntu*

- (a) Open the command prompt *in administrator mode*. To do so, press the Windows key, type “command prompt”, right-click the Command Prompt program, and select “Run as administrator”.
- (b) In the command prompt, type `wsl --install`.
- (c) Restart your computer.

2. *Updating WSL (if you’ve installed it before)*

- (a) If you've installed WSL before, make sure it's updated by running in the Command Prompt `wsl --update`.

3. *Updating Ubuntu*

- (a) Open Ubuntu. Again, you may press the Windows key and type "Ubuntu". Ubuntu may take a few minutes to set up the first time. This is normal.
- (b) Create a new user account with a username and password when prompted. This does not need to correspond to your Windows username. Make sure to remember your password, or skip setting one by just pressing the Enter key instead of entering a password.
- (c) Run the command `sudo apt update` to update the package list.
- (d) Run the command `sudo apt upgrade` to install any necessary updates.

Navigating the WSL File System

When WSL installs a Linux *distribution* (meaning any Linux-based operating system; in our case it's Ubuntu), it creates a home and file system separate from Windows. While you can still access your files in Windows from WSL by navigating through the `/mnt/` directory, we recommend keeping your code in the Linux file system. This is because in WSL 2 (the version of WSL you should now have), accessing files on the Windows file system is *much* slower than accessing files on the Linux file system. (In WSL 1 this was not the case, but WSL 2 made changes that sacrificed speed in accessing Windows to improve performance in other areas.)

When the Ubuntu terminal first starts up, the "current working directory" is your user folder. Its absolute path is `/home/<username>`, but it is abbreviated by the tilde, `~`. At any time you may navigate back here with the command `cd ~`, or even just `cd` for short.

If you'd like to create a new directory in your user folder—say, a "documents" folder in which to keep your ACME code—you can navigate to your user folder, then execute the command `mkdir documents`.

You can configure Ubuntu to automatically navigate to a folder of your choice instead of your user directory `~`. You may follow these steps:

1. Open your Ubuntu terminal.
2. Type `cd ~` to go to your home directory.
3. Type `nano .profile` to open your profile file.
4. Add the path of the desired default directory to the end of the file. For example, if you wanted to change the default "boot" directory to be your `documents` folder, you would add the following line of code to the end of the file:

```
cd ~/documents
```

5. Save and exit the file by pressing `Ctrl+X`, then `Y`, and finally `Enter`.
6. Close your terminal and reopen it.

Ubuntu should now immediately start in your `documents` folder.

Git

Git is a version control system that helps you manage changes to your code over time. This section will serve as a guide for the later steps, so do not worry about learning everything right now. It will come with time. Git allows you to keep track of different versions of your code, collaborate with others, and revert changes if necessary.

To install Git on WSL or another Linux system, run the following command in Ubuntu:

```
sudo apt install git
```

Application in ACME

In ACME, Git is not only used to save and organize your work, but also to turn in assignments. The most current version of your code will be *pulled* by the instructor at a predesignated time and graded. If you always *push* your code when you're done working on it, you will never miss turning in an assignment. Explanations of these Git terms are provided later in this document.

Downloading Course Materials

The lab manuals can be found at <https://acme.byu.edu/>. You can also download the zip folders that contain the lab materials from this page. However, you won't need to download them since the method we will use to set up your GitHub repository will not require them. There is a more involved way to set everything up (found in the Additional Materials section), which involves downloading the zip files and pushing your materials up to a GitHub repository, but the following procedure is much simpler.

Online Setup

1. *Sign up for GitHub.* If you already have a GitHub account, sign into your account and proceed to the next step. Otherwise, create a GitHub account at <https://github.com/>. This is where you will store your files online for grading.
2. *Set up your Git credentials.* You will need to set up your Git `user.name` and `user.email`. `user.name` should be your actual name, and `user.email` must be the email associated with your GitHub account. To do this, open Ubuntu, and run the following code:

```
$ git config --global user.name "your name"  
$ git config --global user.email "your email"
```

3. *Create an ssh key.* This step only needs to be done once on each computer you want to use to access your repository. If you have multiple repositories on the same computer, you do *not* need to repeat this step for each one. To create an ssh key, enter the following command in Ubuntu:

```
ssh-keygen -t ecdsa -b 256
```

Press the **Enter** or **Return** key to accept the default file location. It will then prompt you to enter a Password, but you can press **Enter** or **Return** again to skip this step if you don't want a Password. The key will then be created. The file for the key will be placed in the `/home/<username>/.ssh` (or `~/.ssh`) directory.

Now that the key is created, you need to add it to your GitHub account. From GitHub, click on your profile icon in the upper right corner, and click on **Settings** towards the bottom of the menu. Now click on **SSH and GPG keys** on the left, and click the green **New SSH key** button. Make a Title for this key (it doesn't matter what you put, but you may wish to specify which machine this key will be used for). Make sure the **Key type** is set to **Authentication Key**.

Now, using the file explorer, navigate to the `.ssh` folder, and open the *public key* file. This file should be called `id_ecdsa.pub`; do *NOT* use `id_ecdsa` (without the `.pub` extension). Copy the contents of this file and paste it into the Key field on GitHub.

If you're having trouble navigating to the `.ssh` folder, you can try to print the contents of the folder by running this code in your terminal:

```
cat ~/.ssh/id_ecdsa.pub
```

Once you've pasted the contents of `id_ecdsa.pub` into GitHub, press the green **Add SSH key** button. Then, in your terminal run

```
ssh-add ~/.ssh/id_ecdsa
```

If you get an error that says "Could not open a connection to your authentication agent," then run

```
eval $(ssh-agent)
ssh-add ~/.ssh/id_ecdsa
```

To verify that this worked, when you make your first commit the new ssh key should be added to the file `~/.ssh/known_hosts`.

4. *Clone GitHub Classroom Repository.* In this step, you will set up a GitHub repository with all of your lab materials. The following steps will guide you through the easiest way to accomplish this, in which you will simply clone a template repository from GitHub Classroom.

Repeat this procedure for both Volume 1 and Volume 2.

- On *Learning Suite*, you will find a link to a GitHub Classroom. Open the link in a new tab.
- Click the green **Accept this assignment** button.
- After a few seconds, refresh the page. The page should now say "You're ready to go!" Beneath the line that reads "Your assignment repository has been created:" there should be a GitHub link. Click the link.
- You will be directed to your personal GitHub repository, which you must now clone onto your computer. To clone the repository, click on the green **Code** button. Make sure the **SSH** option is selected, then copy the link below it.

- Now, in Ubuntu, navigate to a good location on your computer where you will want all your lab materials (i.e. **Documents**, **Desktop**, etc.), then type `git clone` followed by a space and then the link you copied from GitHub. It should look something like

```
git clone git@github.com:classroom_name/repo_name.git
```

Then press **Enter** or **Return**. An identical clone of the GitHub repository should now download onto your computer.

5. *Download data files.* Many labs have accompanying data files. To download these files, navigate into each of your course directories and run the `download_data.sh` bash script, which downloads the files and places them in the correct lab folders for you. You can also find individual data files through [Student-Materials/wiki/Lab-Index](#)

```
# Navigate to your folder and run the script
$ cd /path/to/folder
$ bash download_data.sh
```

Repeat this process for both Volume 1 and Volume 2.

Installing Python and Dependencies

To install Python, navigate into one of your course directories (e.g. `Volume1`) and run

```
bash install_python.sh
```

Now restart your terminal.

The next step is to set up a “virtual environment.” Using a separate virtual environment for separate Python projects makes it possible to isolate sets of dependencies from each other. This prevents packages for one project from inadvertently interfering with those of another project—for example, your ACME labs will need one set of packages, while another project you do in Python may require different versions of the same packages. Environments are considered good practice in industry, if not essential; for the small amount of extra effort, they can save a *lot* of headache.

We’ll be using Python’s native virtual environment system, `venv`, and we’ll be setting up just one environment for all our ACME code (since there’s just one set of dependencies). In the terminal, navigate to the folder where you’d like to keep your ACME lab folder(s), for example, `~/documents`. Then to create an environment in a folder named `.venv`, write the command

```
python -m venv .venv
```

To “activate” the environment—that is, to switch to the environment’s version of Python together with its associated dependences—use the `source` command:

```
source .venv/bin/activate
```

NOTE

If you accidentally create the virtual environment folder in the wrong location or with a name you don't like, no worries! Just delete the folder and create a new one with the instructions above. **Do not** just move or rename the folder, as the environment contains scripts that will not get properly updated, and the environment will simply not work.

You should see your activated environment (`.venv`) appear at the beginning of the current line in the terminal, like `(.venv) <username>@<computer_name>:.` To deactivate the environment, simply write `deactivate`. You should see `(.venv)` disappear.

We're finally ready to install some Python packages to our environment. First, **make sure you've activated your new environment** and verify that `(.venv)` appears at the start of the current line of the terminal. Now run

```
bash install_dependencies.sh
```

The last line of the output should say "Python dependencies successfully installed!" If it doesn't, double-check that you've activated your virtual environment.

You have now installed Python and all necessary dependencies you will need in ACME!

Using VS Code

VS Code is the recommended code editor in ACME, though it is not required. We recommend it because it's free and open source, and it can use WSL very easily. You can download it at <https://code.visualstudio.com/> Once you have it installed, you can search for it on your machine under: `Visual Studio Code`. You can also open it from the terminal by typing `code`.

1. *Installing the Remote WSL Extension:* The Remote WSL extension allows you to use VS Code to edit files in WSL. You can install it by opening VS Code and clicking on the **Extensions** tab on the left (little building blocks). Search for **WSL**. Then click install.

You will need to restart VS Code for the extension to take effect.

2. *Opening a Folder in WSL:* In order to open a folder in WSL, press `Ctrl+Shift+P`, then search "Remote: Show Remote Menu" and press `Enter`. Click **Connect to WSL**. Now click on the **File** menu in the upper left, and click **Open Folder**. Type in the complete file path to one of your course directories, such as `/home/<username>/documents/Volume1`. Click **OK** to save this path. Repeat these steps for the other Volumes.
3. *Installing the Python Extension:* Search in the **Extensions** tab for "Python", then click install. This will enable VS Code's support for Python.

It will also make it possible for VS Code to use your virtual environment. Press `Ctrl+Shift+P`, then search "Python: Select Interpreter" and press `Enter`. If the path to your virtual environment doesn't come up, click "Enter interpreter path...", then "Find...", then start by typing the path to your virtual environment folder. Select the Python file inside this folder, located at `<virtual_environment_folder>/bin/python`.

Setup on a Lab Machine

Most ACME students prefer to do their work on their own machines. However, throughout the semester something may happen where you are unable to install something, or your computer might break, and it will be easier to use a different computer. For this reason, it is recommended that you set up your repository on a lab machine as well. The software on the lab machine will always work, and if it somehow doesn't, you will have dedicated IT help with the lab computers.

The simplest way to set up a lab machine is to log into one of them in person, but you can also access one remotely through the *Secure Shell Protocol* (ssh). If you are unfamiliar with the ssh process, don't worry; you will learn this skill and many others shortly in the ACME program. For now, we will explain some ssh basics (or if you have physical access to a lab machine, you may skip this paragraph). To ssh onto a lab machine, open Ubuntu and type:

```
ssh <netID>@acme<number_between_01_and_30>.byu.edu
```

You will then be prompted for your password; this will be your BYU password. If this doesn't work, try another number (it will not matter which number you end up using). When you ssh onto a lab machine, your terminal gains control of the lab computer. You can verify this by checking that the machine name before the \$ is now different than before and says "acme".

Now that you have access to a lab machine, you will need to repeat the step *Create an ssh key* from the *Online Setup* section. This will involve uploading another ssh key to GitHub specific to this lab machine. Next, you will need to clone your repository onto the lab machine. To do this, first navigate into the *myacmeshare* folder by running

```
cd myacmeshare
```

in your terminal. Then, go to GitHub and navigate to the repository you wish to clone. Find and click the green **Code** button. Make sure that **SSH** is underlined and copy the link; then run

```
git clone <link>
```

in your terminal. This will clone your repository onto the lab machine.

You will also need to repeat the step *Download data files* from the *Online Setup* section. Once you have repeated this process for both volumes, you are ready to go. You can now access your repository on the lab machine. Now, if anything goes wrong, you can either ssh onto the lab machine or use it in person, and you will be able to access and work on your ACME labs without a problem.

You can also use VS Code to ssh onto the lab machine in a similar way to connecting to WSL. To do this, install the Remote-SSH extension using the **Extensions** tab on the left (little building blocks) of the window in VS Code. Then, you can add a Remote Window by clicking on the blue **Open a Remote Window** button in the bottom left corner, selecting *Connect To Host*, and typing in `<netID>@acme<number_between_01_and_30>.byu.edu`.

If All Else Fails...

If, while working on a lab, you can't get something to work on both your own computer and the lab machine, your last resort is Google Colab. It should be reserved as a last resort, because Google Colab is formatted in a notebook style, which is fundamentally different than a .py file. However, if you are ever stuck, keep in mind that it's still an option. See the Using Google Colab guide in the Appendix for more information on using Colab for ACME labs.

Additional Git Help

Here are some key concepts and terminology you'll need to understand and use Git effectively. You may need to refer back to this multiple times.

- *Repositories.* A Git repository is a collection of files and folders that Git is tracking. By creating a repository, you are simply telling the software that it should back up certain files that you tell it to. When you create a repository, Git creates a hidden directory called `.git` inside your project folder that tells it to track those files.
- *Adding Files.* Git will only track the files that you specifically add. You can add files individually, or you can add all the files in a folder at once. To add a file, you'll typically use the command:

```
git add <filename>
```

- *Commits.* A commit is a snapshot of your code at a specific point in time. When you make changes to your code, you can create a new commit to record those changes. Each commit has a unique identifier, which allows you to reference it later if you need to revert your code to a previous state. If you are collaborating with others, it gives you an “undo” button specific to each user, allowing you to be more organized. It is good practice to commit every time you leave your computer to keep track of your work.

To create a commit, you'll typically use the command:

```
git commit -m "Commit message"
```

- *Branches.* A branch is a separate line of development that diverges from the main line of development. By creating a new branch, you can work on a feature or bug fix without affecting the main codebase. Once you're done with your changes, you can merge the branch back into the main codebase.

You probably won't branch your code off the main branch in the ACME courses, but this term is still useful to know.

- *Cloning.* Cloning a repository is simply downloading a codebase. Just like downloading a zip folder, all the files are present, but the original author does not have access to your files unless you push them back.

To clone a repository, you'll typically use the command:

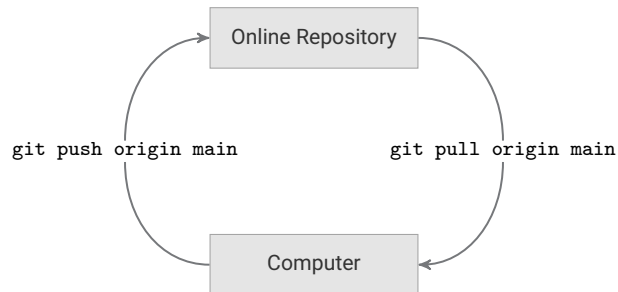
```
git clone <url>
```

- *Pushing and Pulling.* Among the most frequent operations you'll use with Git are pushing and pulling.

When you *push* changes to a remote repository, you're sending your commits to a central server where others can access them. This will be where the grader accesses your code in order to give you a grade. To push your changes, you'll typically use the command:

```
git push <remote> <branch>
```


Here, “remote” refers to the remote repository where you want to push your changes (e.g., “origin”), and “branch” refers to the name of the branch you’re pushing (e.g., “main”). Good practice is to push every few hours of working, and absolutely every time you’re done with a lab. Pushing saves your work on a secure server, meaning if something catastrophic happens to your computer, none of your work will be lost and you can continue where you left off on a different machine. Do not assume you’re different and that computer issues will never happen to you. Doing a lab from scratch twice in a week is guaranteed to take more time than just once!



Exchanging Git commits between the repository and a local clone.

When you *pull* changes from a remote repository, you’re downloading changes that others have made and incorporating them into your local codebase. This will be used to pull your grading feedback back from the grader. To pull changes, you’ll typically use the command:

```
git pull <remote> <branch>
```

Here, “remote” and “branch” have the same meanings as in the `git push` command. Good practice is to pull just before any time you push.

- *Origin and Main.* “Origin” and “main” are two terms you’ll often see when working with Git.

“Origin” refers to the default remote repository where your code is stored. When you clone a repository, Git sets up a remote called “origin” that points to the URL of the original repository. You can push/pull changes to/from “origin” to collaborate with others.

“Main” is the name of the default branch in a Git repository. This is the branch where the main line of development occurs, and it’s typically the branch you’ll be working on most of the time. However, depending on the repository, the default branch may be named something else (e.g., “master”).

Command	Explanation
<code>git status</code>	Display the staging area and untracked changes.
<code>git pull origin main</code>	Pull changes from the online repository.
<code>git push origin main</code>	Push changes to the online repository.
<code>git add <filename(s)></code>	Add a file or files to the staging area.
<code>git add -u</code>	Add all modified, tracked files to the staging area.
<code>git commit -m "<message>"</code>	Save the changes in the staging area with a given message.
<code>git checkout -- <filename></code>	Revert changes to an unstaged file since the last commit.
<code>git reset HEAD -- <filename></code>	Remove a file from the staging area.
<code>git diff <filename></code>	See the changes to an unstaged file since the last commit.
<code>git diff --cached <filename></code>	See the changes to a staged file since the last commit.
<code>git config --local <option></code>	Record your credentials (<code>user.name</code> , <code>user.email</code> , etc.).

Common Git commands.

Now that you've learned about some of Git's features, it's time to implement them in your first ACME Lab, *Intro to GitHub!*

Additional Material

Manual GitHub Setup

Downloading Course Materials

The lab manuals can be found at <https://acme.byu.edu/>. If you are a Junior, you should also download the Volume 1 and 2 zip folders. You should then unzip these in a folder you are familiar with, such as your Documents or Desktop folder.

ACHTUNG!

Make absolutely sure your unzipped lab folders are not nested! When you open each lab folder, you should NOT have to open another folder to access all the lab materials! If your folder is nested, move the inner folder out so that there's only "one layer" to access your lab materials.

Online Setup With Lab Folders

1. *Sign up for GitHub.* If you already have a GitHub account, sign into your account and proceed to the next step. Otherwise, create a GitHub account at <https://github.com/>. This is where you will store your files online for grading.
2. *Create an ssh key.* This step only needs to be done once on each computer you want to use to access your repository. If you have multiple repositories on the same computer, you do *not* need to repeat this step for each one. To create an ssh key, enter the following command in Ubuntu:

```
ssh-keygen -t ecdsa -b 256
```

Press the **Enter** or **Return** key to accept the default file location. It will then prompt you to enter a Password, but you can press **Enter** or **Return** again to skip this step if you don't want a Password. The key will then be created. The file for the key will be placed in the `/home/<username>/.ssh` (or `~/.ssh`) directory.

Now that the key is created, you need to add it to your GitHub account. From GitHub, click on your profile icon in the upper right corner, and click on **Settings** towards the bottom of the menu. Now click on **SSH and GPG keys** on the left, and click the green **New SSH key** button. Make a Title for this key (it doesn't matter what you put, but you may wish to specify which machine this key will be used for). Make sure the **Key type** is set to **Authentication Key**.

Now, using the file explorer, navigate to the `.ssh` folder, and open the *public key* file. This file should be called `id_ecdsa.pub`; do *NOT* use `id_ecdsa` (without the `.pub` extension). Copy the contents of this file and paste it into the **Key** field on GitHub.

If you're having trouble navigating to the `.ssh` folder, you can try to print the contents of the folder by running this code in your terminal:

```
cat ~/.ssh/id_ecdsa.pub
```

Once you've pasted the contents of `id_ecdsa.pub` into GitHub, press the green **Add SSH key** button. Then, in your terminal run

```
ssh-add ~/.ssh/id_ecdsa
```

If you get an error that says "Could not open a connection to your authentication agent," then run

```
eval $(ssh-agent)
ssh-add ~/.ssh/id_ecdsa
```

To verify that this worked, when you make your first commit the new ssh key should be added to the file `~/.ssh/known_hosts`.

3. *Make a new repository.* You will need to create a new repository for both Volume 1 and Volume 2. Note: you *must* follow these instructions exactly. If you do not complete each step exactly as specified below, delete the repository and start over.

- On your GitHub Home page, click the green **Create repository** button to the upper left of the screen; or, if you have already used GitHub before and already have repositories set up, click on the green **New** button next to **Top Repositories**.
- First you will need to provide a Repository name; this can be anything you want, but please make it relevant to the ACME Volume it will represent; you may also provide a repository description if you like.
- Mark the repository as **Private**.
- Leave the box next to **Add a README file** unchecked (if you accidentally include a README, delete the repository and start over).
- Under **Add .gitignore**, select **None** for **.gitignore template** (if you accidentally include a **.gitignore**, delete the repository and start over).
- Under **Choose a license**, select **None** for **License**.
- Finally, click the green **Create repository** button.

Repeat this process for both Volume 1 and Volume 2.

4. *Give the instructor access to your repository (First Day of Class).* If you are doing this before the first day of class, skip to the next step. In your newly created repository, click the **Settings** tab along the top of the page and click on **Collaborators** on the left. Then click on the green **Add people** button. Type your instructor's GitHub username, select them, then click the big green **Add <username> to this repository** button.
5. *Connect your folder to the new repository.* In Ubuntu enter the following commands.

```
# Navigate to your folder.
$ cd /path/to/folder # cd means 'change directory'.

# Make sure you are in the right place.
$ pwd                # pwd means 'print working directory'.
/path/to/folder
$ ls *.md            # ls means 'list files'.
```

```

README.md          # This means README.md is in the working directory.

# Connect this folder to the online repository.
$ git init
$ git remote add origin git@github.com:<username>/<repo>.git
# Make sure the link has this form. If it starts with https, select the ←
  ssh option on GitHub instead.

# Record your credentials. user.name should be your name, and user.email ←
  must be the email associated with your GitHub account.
$ git config --global user.name "your name"
$ git config --global user.email "your email"

# Add the contents of this folder to Git and update the repository.
$ git add --all
$ git commit -m "initial commit"
$ git branch -M main    # Update branch name.
$ git push origin main

```

For example, if your GitHub username is greek314, the repository is called acmev1, and the folder is called Student-Materials/ and is located in Desktop, you would enter the following commands:

```

# Navigate to the folder.
$ cd ~/Desktop/Student-Materials

# Make sure this is the right place.
$ pwd
/Users/Archimedes/Desktop/Student-Materials
$ ls *.md
README.md

$ git init
$ git remote add origin git@github.com:greek314/acmev1.git

$ git config --global user.name "archimedes"
$ git config --global user.email "greek314@example.com"

$ git add --all
$ git commit -m "initial commit"
$ git branch -M main
$ git push origin main

```

At this point you should be able to see the same files on your GitHub repository that are also found on your machine in your course directory. If you enter the repository URL incorrectly in the `git remote add origin` step, you can reset it with the following line:

```
$ git remote set-url origin git@github.com:<username>/<repo>.git
```

NOTE

You may get an error like the following when you run `git push`:

```
...
fatal: Authentication failed for 'https://github.com/<username>/<repo>
.git/'
```

If this error occurs, your repository URL is in the wrong format; most likely, you used the `https` format instead of the `ssh` format shown above. You can use the `git remote set-url origin` command to fix this issue as well.

6. *Download data files.* Many labs have accompanying data files. To download these files, navigate into each of your course directories and run the `download_data.sh` bash script, which downloads the files and places them in the correct lab folders for you. You can also find individual data files through [Student-Materials/wiki/Lab-Index](#)

```
# Navigate to your folder and run the script
$ cd /path/to/folder
$ bash download_data.sh
```

Repeat this process for both Volume 1 and Volume 2.