



# Getting Started On Windows

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

## Installing WSL

Windows Subsystem for Linux (WSL) is a compatibility layer for running Linux natively on Windows 10 and 11. We'll go through the steps to install Ubuntu on Windows using WSL, which you will use to 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. *Enabling WSL:*
  - (a) Open the Start menu and search for "Turn Windows features on or off".
  - (b) Click on it.
  - (c) In the "Windows Features" window, scroll down until you see "Windows Subsystem for Linux" in the list.
  - (d) Check the box next to it to enable the feature.
  - (e) You should also enable "Virtual Machine Platform".
  - (f) Click "OK" to save your changes.
2. *Installing Ubuntu:* Now that WSL is enabled, you can install Ubuntu on your computer.
  - (a) Open the Microsoft Store app on your computer.
  - (b) Search for "Ubuntu" in the search bar and select "Ubuntu" from the search results.
  - (c) Click the "Get" button to download and install Ubuntu on your computer.
  - (d) Wait for the installation to complete.
3. *Launching Ubuntu:* Now that Ubuntu is installed, we can launch it and start using it.
  - (a) Press the Windows key and type "Ubuntu" in the search bar.
  - (b) Select "Ubuntu" from the search results to launch it.

- (c) Wait for Ubuntu to start up and create a new user account with a username and password when prompted.

Make sure not to forget your password!

#### 4. *Installing Linux Updates:*

- (a) In the Ubuntu terminal, run the command `sudo apt update` to update the package list.
- (b) Run the command `sudo apt upgrade` to install any necessary updates.

## Finding Your Files in WSL

WSL has its own independent home and file system apart from Windows, but you can still access your files in Windows from WSL by navigating back through the `mnt` directory. For example, to access your `c` drive (your files) you can use `cd /mnt/c` to navigate to your standard head directory. So if you want to access your `Desktop`, you can find it at `/mnt/c/Users/<username>/Desktop`. This may feel counterintuitive to how Windows normally works, but this one difference is a small price to pay to make your life much easier.

Fortunately, you can actually change the default “boot” directory that Ubuntu starts in. The process for changing your boot directory is very simple, and it is recommended that you do so to make your life easier.

1. Open your Ubuntu terminal.
2. Type `cd ~` to go to your home directory.
3. Type `sudo 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 base user directory, you would add the following line of code to the end of the file:

```
cd /mnt/c/Users/<username>
```

where `<username>` is the name of your user.

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 user directory.

## 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 your `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, you should see your new ssh key in 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.

## Using VSCode

VSCode is the recommended code editor in ACME, though it is not required. We recommended 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 VSCode to edit files in WSL. You can install it by opening VSCode and clicking on the **Extensions** tab on the left (little building blocks). Search for WSL. Then click install.

You will need to restart VSCode for the extension to take effect.

2. *Opening a Folder in WSL:* In order to open a folder in WSL, click on the bottom left corner of VSCode (blue box) to Open a Remote Window. Then click **Connect to WSL**. Now click on the **Explorer** icon in the upper left (two pages), and click **Open Folder**. If you see

```
/home/<username>
```

in the searchbar, you are accessing the WSL home directory, not the one where your files are. Type in the complete file path to one of your course directories, such as `/mnt/c/Users/<username>/Desktop/Volume1`. Make sure to navigate into the folder that contains all the lab materials, not just `Desktop` or another higher directory. Click `ok` to save this path. Repeat these steps for the other Volume.

## Installing Python

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

```
bash install_python.sh
```

Now restart your terminal. On Windows or Linux, you will need to run the following commands in WSL or Standard Linux Terminal:

```
sudo apt-get install graphviz
sudo apt-get install libopenmpi-dev
sudo apt-get install ffmpeg
sudo apt-get install g++
sudo apt-get install python3-tk
sudo apt-get install openjdk-8-jdk
```

Assuming no errors, you should now run

```
bash install_dependencies.sh
```

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

## 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 VSCode 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 VSCode. 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:

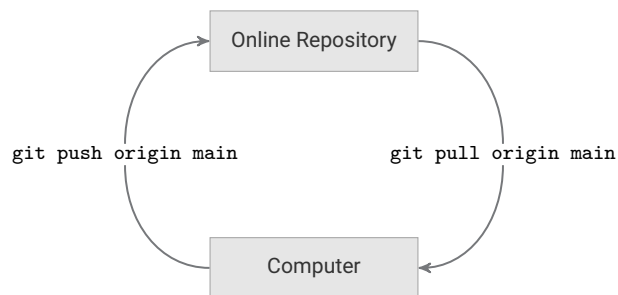
```
git clone <url>
```

- *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 &lt;filename(s)&gt;</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 "&lt;message&gt;"</code>	Save the changes in the staging area with a given message.
<code>git checkout -- &lt;filename&gt;</code>	Revert changes to an unstaged file since the last commit.
<code>git reset HEAD -- &lt;filename&gt;</code>	Remove a file from the staging area.
<code>git diff &lt;filename&gt;</code>	See the changes to an unstaged file since the last commit.
<code>git diff --cached &lt;filename&gt;</code>	See the changes to a staged file since the last commit.
<code>git config --local &lt;option&gt;</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, you should see your new ssh key in 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"

# Update branch name.
$ git branch -M main

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

$ git add --all
$ git commit -m "initial commit"
$ 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.





# Using Google Colab

**Lab Objective:** *Google Colab is an environment similar to Jupyter Notebooks that is run remotely on Google's servers. It has some advantages over other environments, including easy package management and installation and the ability to run on a machine that doesn't have python installed. This appendix details how Colab can be used for ACME labs, as well as some issues to be aware of and relevant workarounds.*

## Setup

Google Colab can be started by going to the Colab website <https://colab.research.google.com/> or by opening a `.ipynb` file from your Google Drive. When Colab starts, you can upload a `.ipynb` file by selecting the **Upload** tab and choosing the `.ipynb` file you wish to upload. Colab can also be used with `.py` files, but the process is somewhat more involved and is detailed below.

Once your file is open in Colab, it can be run and edited like a normal Jupyter notebook. To submit your lab for grading, you will need to download the file by selecting **File > Download** and then either **Download .ipynb** or **Download .py**. You will then need to move your file into the repository clone on your machine in the correct lab folder, where you can then add, commit, and push it like normal.

### ACHTUNG!

Remember to keep the name of the file ***IDENTICAL*** to the original name of the lab spec! Failure to do this will cause the test driver to skip over your file and give you an automatic zero for the lab!

## Using `.py` Files with Colab

Google Colab only works with `.ipynb` files, but many ACME labs use `.py` files. Since the former filetype has additional formatting, there isn't a way to directly import a `.py` file into Colab. The simplest way around this is to create a new file in Colab and copy-paste the lab file's contents into it. This way, it can be edited and used as a normal Jupyter notebook. Then, when you're finished, download the notebook as a `.py` file instead of a `.ipynb` file.

However, under these circumstances some care must be taken in order for your work to be graded properly. In particular, before you download your file, make sure that the following are satisfied:

- All Colab-specific code to upload files is commented out; as the packages used do not exist outside of Google Colab, the test driver will raise an error when it attempts to import them.
- There are no calls to the lab's functions, for testing purposes or otherwise, outside of a typical `if __name__=="__main__"` statement; otherwise, they will be run when the file is imported by the test driver, which will certainly cause issues.

## Using Data Files

To use a data file in Colab, you must first upload it by running the following in a code cell in Colab:

```
from google.colab import files
files.upload()
```

When run, this will create a prompt for you to upload your files. After doing so, the data files can be accessed from the notebook as normal.

The best way to upload an entire folder is to first zip it, then upload the zipped folder to Colab, and then unzip it in the Colab notebook:

```
# To upload a folder called 'data', first zip it, and then upload 'data.zip'
files.upload()
# Then, it can be unzipped
!unzip data.zip
```

If the files do not need to be in the same folder, you can always simply upload multiple files at once with the usual `files.upload()` code.

## Installing Packages

One advantage of Google Colab is that a very large percentage of the python packages used in the ACME labs are already pre-installed. However, there are a few packages that are not installed, and must be installed each session. Packages can be installed by running `pip` in a code cell, which is done by putting an exclamation point before the usual command. For example, to install the GeoPandas package, you would run:

```
!pip install geopandas
```