B. Installing Necessary Software

B.1. For Microsoft Windows

Windows 10:

In August of 2016 Microsoft released an update for Windows 10 that allows you to install a complete development environment, including the tools used in this book, fairly easily.

1. To begin, see Microsoft's guide to installing "Bash and Ubuntu on Windows":

   https://msdn.microsoft.com/en-us/commandline/wsl/install_guide

2. Once you've followed those instructions, type the following commands in the `bash` window:

   ```
   sudo apt-get update
   sudo apt-get -y install g++ nano gnuplot-x11
   ```

   This will install the specific software (*g++*, *nano*, and *gnuplot*) used in this book.

3. Next, type the following command:

   ```
   echo export DISPLAY=localhost:0 >> $HOME/.bashrc
   ```

4. In order to use this version of *gnuplot* you'll need to install one more piece of Windows software, called an "X server". This allows the tools in the development environment (installed in steps 1 and 2) to display graphics on your screen. To install it, download and install VcXsrv from here:

   https://sourceforge.net/projects/vcxsrv/

5. After you've installed VcXsrv, hold down the "Windows" key and type R, then type:

   ```
   c:\program files\vcxsrv\xlaunch.exe
   ```

   and press the Enter key. A window like Figure B.1 should appear. Keep clicking "Next" until you get to the dialog box shown in Figure B.2.

   Click "Save Configuration" and save the configuration as `config.xlaunch` on your desktop.

   Now hold down the "Windows" key and type R, then type:

   ```
   shell:startup
   ```

   and press the Enter key. This will open up your startup folder. Drag the `config.xlaunch` icon from your desktop into this folder.

6. Now restart your computer. You should be able to get a command window by clicking "Bash on Ubuntu on Windows" in the Start Menu.

Windows 7 and 8:

For Windows 7 and 8 we've created a bundle of useful free Windows software that you can download and install on your own computer. The bundle includes *g++*, *nano*, and *gnuplot*, among other tools.

You can download the bundle from the following address:

http://faculty.virginia.edu/comp-phys/phys1660/2015/software/phys1660-bundle-setup.exe

The downloaded file will be called `phys1660-bundle.exe`. Run it to install the software. Once installed, you should see a new icon for *MSys* on your desktop. Double-click this to open a command window. From the command window, you can use the *g++*, *nano*, and *gnuplot* commands described in this book.

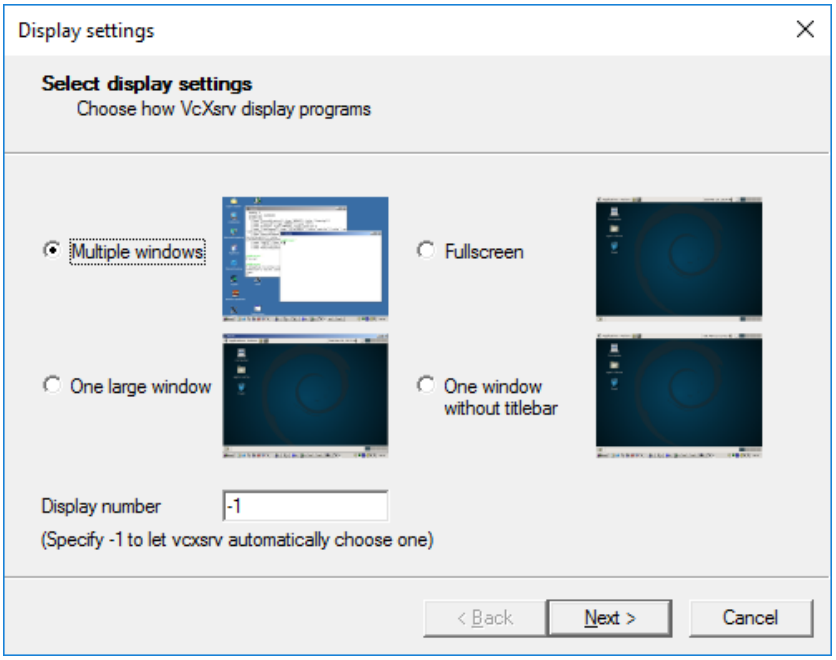

Figure B.1: Running `xlaunch`.

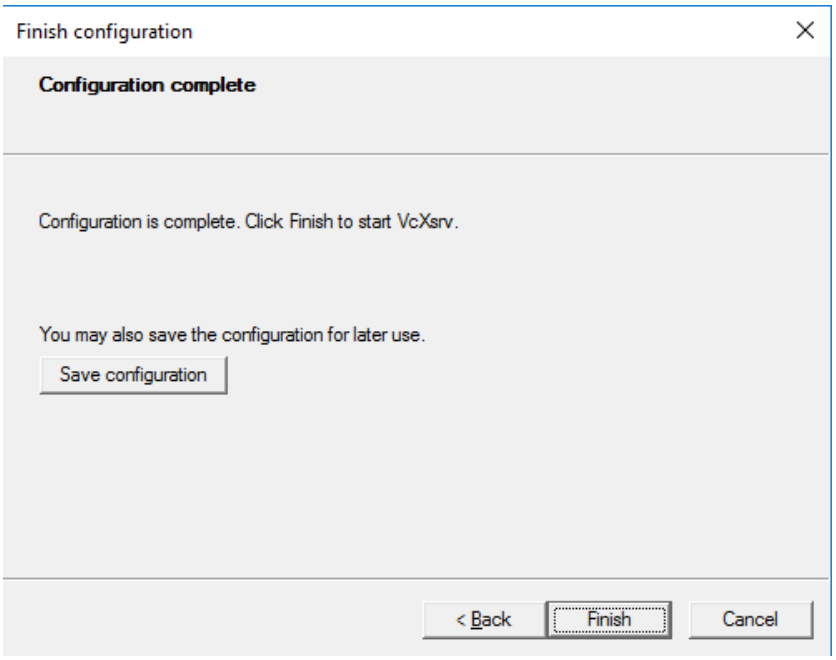

Figure B.2: Saving `xlaunch` configuration.

B.2. For Linux

On Debian, Ubuntu, Mint and similar distributions, type:

```
sudo apt-get update
sudo apt-get install g++ nano gnuplot
```

On Fedora, CentOS, Red Hat and similar distributions, type:

```
sudo yum install gcc-c++ nano gnuplot
```

B.3. For Apple OS X

If you're a Mac user, you can install the *Xcode* package from Apple to get *g++* and *nano*. You can get *Xcode* from the Apple Web site (I recommend you **NOT** get the beta version). Here's the address:

https://developer.apple.com/xcode/downloads/

To get a command window, click any blank spot on your desktop background, then go to the "Go" menu at the top of the screen and select Utilities->Terminal.

In order to use *gnuplot* under OS X, you'll also need to install two more things. The first is *XQuartz*, which you can get here:

http://xquartz.macosforge.org/landing/

After you've installed *XQuartz*, you must log out of your computer and log back in to complete the installation. (If you don't do this, gnuplot may not install or work correctly.)

The last thing to install is *gnuplot* itself, which you can get here:

http://faculty.virginia.edu/comp-phys/phys1660/2015/software/gnuplot-4.2.5-i386.dmg

After you've downloaded the file, double-click on it, then drag the *gnuplot* icon onto the Applications icon to install *gnuplot*.

It would be a good idea to also install the custom gnuplot configuration file you can download here:

http://faculty.virginia.edu/comp-phys/phys1660/2015/software/dot.gnuplot

After downloading the file, open a command window and type:

```
mv ~/Downloads/dot.gnuplot ~/.gnuplot
```

(If you don't do this, you'll need to type "`set term x11`" each time you start gnuplot.)

Note: The first time you run gnuplot, you may need to press the "Control" key while clicking, then click "Open". This is because your computer doesn't trust the newly-downloaded application from an unknown source.