ELE 206/COS 306: Software Installation Guide

You will be installing two required programs for Verilog development in this class. The first, Icarus Verilog, is a widely-used Verilog compiler. The scripts that we will provide to compile simulations of your code use Icarus Verilog. The second, GTKWave, is a simple and easy-to-use waveform viewer. You'll be using it in the labs to view the results of your simulations and debug your Verilog code. In later labs, Xilinx's Vivado Design Suite will be used to program FGPA boards using the lab computers in the undergraduate ELE lab.

In addition, we strongly recommend the use of Visual Studio Code for code development. VSCode is a free open-source IDE that can perform syntax highlighting on a variety of programming languages, including Verilog. We recommend it because it is fast, cross-platform, and contains an integrated terminal which allows Verilog commands to be easily executed. All of the examples/tutorials in this course will use VSCode.

Both Icarus Verilog and GTKWave are free open-source software targeted at Unix-like operating systems such as Linux and OS X, but are also available for Windows. Both programs must be used from the command line. The computers in the undergraduate ELE lab have both programs installed, and you may use them there.

We have specific installation instructions for Windows, Mac OS X, and for Debian-based Linux distributions (such as Ubuntu). If you have difficulty getting these installations to work, contact the lab teaching assistants for help. Finally, if you'd prefer to install these packages from source on OS X or Linux, the TA's can help with that as well.

Windows Instructions

Installing VSCode

The easy-to-install VSCode package is available at https://code.visualstudio.com/. Navigate there and download the latest version. Go through the installation and accept the license agreement. Configure the optional settings as desired.

Now open the application (which should be in your Start Menu) and click on the last icon on the left-most bar. This should bring up the extensions toolbar. Search for Verilog HDL and click install. This will enable syntax highlighting for Verilog in VSCode. You can open an integrated terminal in VSCode by navigating to View→Terminal.

Installing Icarus Verilog and GTKWave

An easy-to-install package containing Icarus Verilog and GTKWave is available at http://bleyer.org/icarus/. At the time of this writing, the most recent stable release of Icarus Verilog is available as the first item in the bulleted list under the "Download" section: iverilog-10.1.1-x64_setup.exe [9.77MB]

Download this installer. You must then right click on it and select "Run as administrator." If do not run the installer as administrator, it will run without notifying you of any problems, but fail to add the locations of the executables to the system PATH at the end. Follow the installation wizard. You must accept the GNU GPL agreement, and read the message on the next screen about restrictions on installation directories. On the next screen, you may choose an installation directory. It should be fine to use the default installation directory. On the next screen, make sure to select "Full installation," which includes GTKWave and libraries needed for Icarus Verilog to run on Windows. The next two screens allow you to create entries for Icarus Verilog in your Start menu and on your desktop. Then click "Install." Once the installation completes, you must make sure to select both check boxes to add Icarus Verilog and GTKWave to your system's PATH before clicking "Finish."

Setting up command prompt

You will need to set up a command prompt in order to run commands. If you are using VSCode, you can open the integrated terminal by navigating to View→Terminal. This will allow you to easily interact with the Verilog compiler and GTKWave and will obviate the need for a separate command line window.

The Windows Command Prompt can also be used to run Icarus Verilog and GTKWave. In every version of Windows since Vista, it may be started by hitting the Windows Start key, typing `cmd.exe`, and hitting Enter. It may also be started by opening the run dialog using the shortcut Win+R, typing `cmd.exe`, and hitting Enter. The Command Prompt may also be found in the Start menu under "Accessories" or "Windows System" depending on your Windows version. You may wish to create a shortcut on your desktop or pin Command Prompt to your Start menu or taskbar for easier access in the future if you are not using VSCode.

Verifying your Installation

To verify that Icarus Verilog is installed, open your command line (either VSCode or cmd.exe), then type the command

```
1 iverilog -V
```

and hit Enter. `iverilog` is the Icarus Verilog compiler. The argument `-V` tells it to just print some version information and exit without doing any work. This should cause some version and license information to be displayed.

Next, type the command

```
1 gtkwave --version
```

Again, the `--version` argument tells `gtkwave` to exit without doing anything. This command will print out some version information.

Icarus Verilog and GTKWave both must be used through a command line. The desktop and Start Menu shortcuts you may have created during installation will only help you navigate to the installation directory. Inside the installation directory, you will find the uninstaller, as well as three PDF files: *iverilog.pdf*, *iverilog-vpi.pdf*, and *vvp.pdf*. These are the manual pages for the three commands that Icarus Verilog provides, which document the arguments that may be given to these commands and their functions.

For a more Linux-like full-featured command line, you may be interested in installing and using Cygwin instead of the Windows Command Prompt.

Mac OS X Instructions

For OS X, the easiest way to install new software distributions is with a package management system. We recommend using Homebrew, a very popular package management system. If you already have Xcode and Homebrew

installed, skip ahead to the section on Brew commands. Otherwise, follow through the full list of instructions.

Installing VSCode

VSCode is available at https://code.visualstudio.com/. Download the latest Mac executable and drag it to your Applications folder. Open Spotlight and type "vsc" to open the app.

Once VSCode opens, click on the last icon on the left-most bar. This should bring up the extensions toolbar. Search for Verilog HDL and click install. This will enable syntax highlighting for Verilog in VSCode. You can open an integrated terminal in VSCode by navigating to View→Terminal.

Installing Xcode

You'll need the C and C++ compilers for Mac OS X to use Icarus Verilog. The best way to get these is to install Xcode, Apple's development environment. Just download Xcode via the Mac App Store and follow their instructions. You won't be developing for iOS, so no need to set up a developer account. Alternatively, Xcode is available on Apple's developer pages, which you can find at https://developer.apple.com/xcode/downloads/. Xcode 7.0 and later should be fine for our purposes.

Once Xcode is installed, open your terminal (open Spotlight and search for "Terminal") and type in the following command:

```
1 xcode-select --install
```

This should install the command-line tools you'll need to compile and install Icarus Verilog and GTKWave.

Installing Homebrew

Next, you should install Homebrew. This will involve using a terminal. If you've never used a terminal, don't sweat it! This is mostly just an exercise in copy-pasting.

Open your terminal (open Spotlight and search for "Terminal") and type in the following command:

```
1 ruby -e "$(curl -fsSL https://raw.githubusercontent.com/Homebrew/install/
    master/install)"
```

Leave your terminal open until the installation completes.

Brew Commands

Now that Homebrew is installed on your system, open your Terminal and type in the following commands:

```
brew tap caskroom/cask
brew update
brew install icarus-verilog
brew cask install gtkwave
echo 'alias gtkwave=/Applications/gtkwave.app/Contents/MacOS/gtkwave-bin' >>
    ~/.bash_profile
source ~/.bash_profile
```

Let each command run to completion. If you see any errors, please ensure that you have Xcode's command-line tool suite installed by running this command:

```
xcode-select --install
```

Congratulations, you've now installed Icarus Verilog and GTKWave! To confirm this, simply type the following instructions into your terminal:

```
iverilog -V
gtkwave --version
```

Each command should print the version number for the respective program. If there's an error, contact a TA for assistance.

Debian-based Linux Instructions

Installing VSCode

VSCode is available at https://code.visualstudio.com/. Download the latest .deb file and open it. You can do this through your OS's file manager/-software manager or via command line with:

```
sudo apt install ./name.deb
```

You can open VSCode by typing it into the search bar in Ubuntu's application menu. Alternatively, you can type `code` in a terminal.

Once VSCode opens, click on the last icon on the left-most bar. This should bring up the extensions toolbar. Search for Verilog HDL and click install. This will enable syntax highlighting for Verilog in VSCode. You can open an integrated terminal in VSCode by navigating to View→Terminal.

Installing Icarus Verilog and GTKWave

For Debian-based Linux distributions, installation is easy using APT.

Simply enter the following two commands into a terminal:

```
sudo apt-get update
sudo apt-get install verilog
sudo apt-get install gtkwave
```

On newer versions of Linux, the apt-get command may have been replaced with apt:

```
sudo apt install verilog
sudo apt install gtkwave
```

Let each command run to completion, entering your administrator password if necessary. Congratulations, you've now installed Icarus Verilog and GTKWave! To confirm this, simply type the following instructions into your terminal:

```
iverilog -V
gtkwave --version
```

Each command should print the version number for the respective program. If there's an error, contact a TA for assistance.

Installing from Source on OS X or Linux

If you'd prefer to install Icarus Verilog and GTKWave from source, you can, although we strongly recommend that you use a package manager. Here's how to install from source. When running the configure scripts and building the source, you may find you are missing dependencies that you may need to install, such as GNU Flex, GNU gperf, GTK+ 2.0 libraries, liblzma, and Tcl/Tk libraries. You can likely install these from a package manager on your system, or build and install them from source found online. Feel free to ask a question on Piazza if you would like help with this.

Build and Install Icarus Verilog

```
wget ftp://icarus.com/pub/eda/verilog/v10/verilog-10.0.tar.gz
tar xzf verilog-10.0.tar.gz
cd verilog-10.0
./configure
make
```

```
sudo make install
cd ..
rm -rf verilog-10.0*
```

Build and Install GTKWave

```
wget http://gtkwave.sourceforge.net/gtkwave-3.3.66.tar.gz
tar xzf gtkwave-3.3.66.tar.gz
cd gtkwave-3.3.66
./configure UPDATE_DESKTOP_DATABASE=/bin/echo
make
sudo make install
cd ..
rm -rf gtkwave-3.3.66*
```

Common Installation Pitfalls

Here are some questions and issues that commonly arise during installation.

Windows

- "After the installer completed, I am trying to verify installation in the command prompt using the commands:

```
iverilog -V
gtkwave --version
```

  However, I keep getting an error that says that the command is not recognized as an internal or external command."

  Suggestion: The installation process should add the `iverilog`, `vvp`, and `gtkwave` commands to your PATH so they are accessible from anywhere. Try closing the command prompt and try re-running the installer and make sure you run it by right clicking on it and select "Run as Administrator". Then, try restarting command prompt and verify installation.

Mac OS X

- "I see an error that looks like this:

```
configure: error: cannot run C compiled programs.
If you meant to cross compile, use '--host'.
```

I am not sure what this means?"

Suggestion: You may not have Xcode's command line tools installed; this suite provides compilers and other useful things that are needed to install Icarus Verilog and GTKWave. To fix this, run the `xcode-select --install` command.

- "When installing Homebrew from the terminal, the terminal prompts me to enter my password, but then is unresponsive."

  Suggestion: It's conventional for command line tools that require a password to not show anything when you're typing your password, so someone looking over your shoulder doesn't learn about your password. Just type your password and hit Return, and it should proceed.

- "I get an error: 400 Bad Request after running the command to install Homebrew:

```
1 ruby -e "$(curl -fsSL https://raw.githubusercontent.com/Homebrew/install
    / master/install)"
```

  Why might that be?"

  Suggestion: Be careful when copying directly from the pdf, sometimes there may be spaces where there shouldn't be!

- "When I run the `brew install icarus` command, I get an error message `Error: No available formula with the name "icarus"`."

  Suggestion: Make sure to type `brew install icarus-verilog` exactly as it appears here and in the lab manual. No extra spaces and do not forget the hyphen!

- "When I run the `brew install icarus-verilog` command, I get an error message `Error: No available formula with the name "icarus-verilog"`."

  Suggestion: This may mean your brew is not up-to-date. Please use `brew update` first to make sure it is up-to-date, then re-run the install commands.

- "When I run the `brew install homebrew/gui/gtkwave` command, I get an error message `Error: No available formula with the name "homebrew/gui/gtkwave"`."

  Suggestion: This may mean your brew is not up-to-date. Please use `brew update` first to make sure it is up-to-date, then re-run the install commands.

- "At the end of an install, I receive an error similar to `Error: The 'brew link' step did not complete successfully`."

  Suggestion: This typically means that there are some file permission issues keeping homebrew from linking. To fix this, run the following command:

```
sudo chown -R $'(your computer username)' /usr/local/lib
```

  Afterwards, you will want to use the link command to link any installed, unlinked packages. An example would be:

```
brew link icarus-verilog
```

- "When I start gtkwave, errors appear in the terminal. Does this mean something is wrong?"

  Suggestion: If you are able to see waveforms and navigate gtkwave, it should be safe to ignore these warnings.

- "When I run `gtkwave filename.vcd` command to see waveforms of the recorded data in .vcd file named *filename.vcd*, I get an error message and cannot see waveforms or open gtkwave.

  Suggestion: Run the following command to see waveforms of the recorded data in *filename.vcd*:

```
open -a gtkwave filename.vcd
```

Linux

- "When I try installing VSCode through Ubuntu's software manager, the installation starts but then quickly ends without finishing the installation. What should I do?"

  Suggestion: Try installing the package via command line instead of through the software manager.