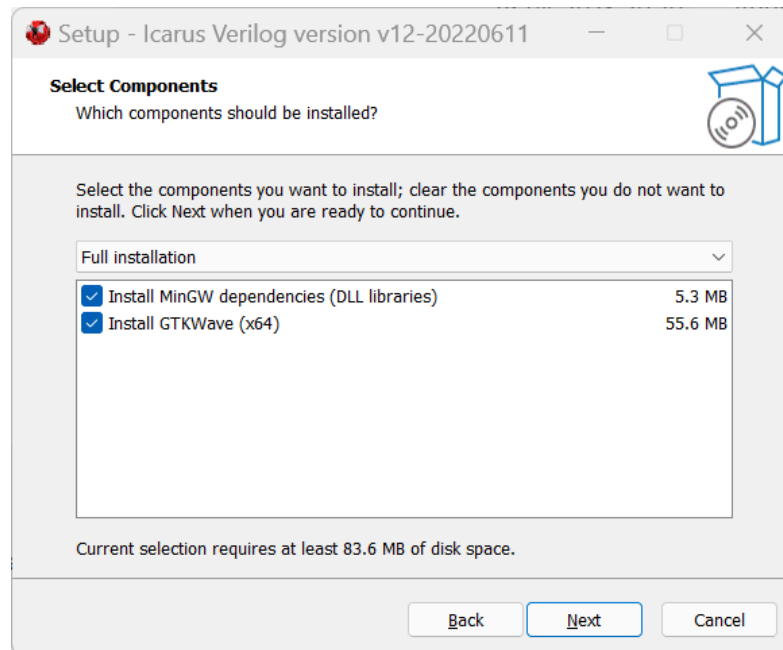


iVerilog installation and usage manual

Windows

- Download link: <https://bleyer.org/icarus/>
- Install the latest version of iverilog (iverilog-v12-20220611-x64_setup).



- Install GTKWave and MinGW dependencies in the setup window.

Linux

Guide

https://iverilog.fandom.com/wiki/Installation_Guide#Installation_From_Premade_Packages

There is no need to compile iverilog from source. It can be installed directly from repositories using apt-get/yum commands. In addition to iverilog, gtkwave also needs to be installed.

For Ubuntu

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

Mac OS

Guide

<https://ee.sonoma.edu/resources/computer-hardware-and-software/icarus-and-gtkwave-mac>

Commands to run iverilog

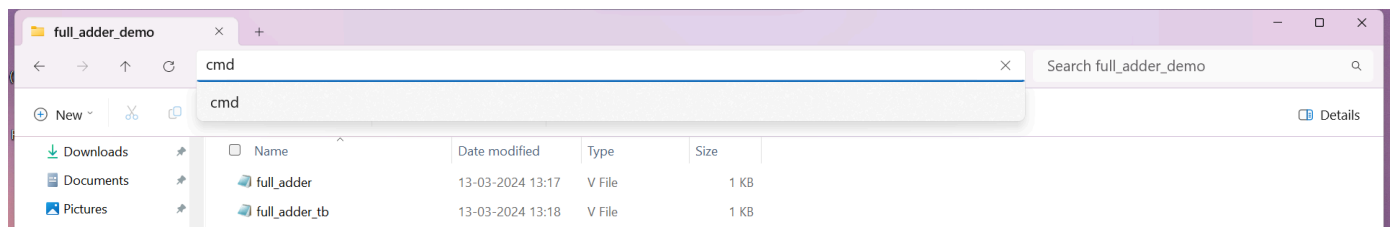
Create a folder VLSI-workshop and copy paste the full_adder_demo folder. Navigate to the full_adder_demo folder and open terminal/command prompt

For Linux users

Right click and open in terminal

For Windows users

Type cmd in the address bar inside full_adder_demo folder



Type the following commands to run Verilog simulation

Example for full adder

```
$ iverilog -o fa.out full_adder.v full_adder_tb.v
$ vvp fa.out
$ gtkwave dump.vcd
```

Generic commands (Modify as per your file names)

Compile Verilog design and test bench files

```
$ iverilog -o <output_file.out> <input files>
```

Run the simulation

```
$ vvp <output_file.out>
```

View the waveform

```
$ gtkwave dump.vcd
```