Setting up iverilog and GTKWave on local machine

Icarus Verilog is a command-line compiler for Verilog HDL. Typically a Verilog module is written for the hardware component (like for a AND-OR gate combination circuit with 3 input and one output defined in this module) and a testbench is written to give various inputs at time intervals and obtain a waveform of output signals. For testing, you should be able to test all permutations of input signals to get the desired outputs.
For multiple files, there is usually a top-level module that uses all the other modules and a testbench is defined for the top-level module.

GTKWave is a waveform visualizer that lets you examine the input and output signal states at various time instants.

You only need to write <component.v> and <component_testbench.v> files. The VCD dump should be generated by the component_testbench file.

Installing iverilog:

- On Ubuntu:

```
sudo apt-get install iverilog
```

Installing GTKWave:

- On Ubuntu:

```
sudo apt-get install gtkwave
```

Testing your installation and learning to run Verilog modules:

Command format to compile testbench and the verilog module:

```
iverilog -o <output> <testbench> <code>
```

For example, if I want to run the test code:

```
iverilog -o test_tb test_tb.v test.v
```

Execute the compiled file:

```
vvp test_tb
```

To open GTKWave:

```
gtkwave test_tb.vcd
```

- Select `test_tb` on the left.
- You should see a list of signals and types as defined in the testbench.
- **Hold and drag** each signal to the waveform window (the black area to the right)

Go to `Time > Zoom > Zoom Best Fit` to see the whole waveform.