

Verilog Installation Procedure

You will Required to download two software , Icarus verilog and GTKwave. The installation procedure can be found below. People who are finding it difficult to run verilog codes in their local system can also use the online EDA platform called EDA playground which is also very convenient.

Installation:

For Debian based operating systems like Ubuntu:

- To install iverilog:

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

- To install GTKWave:

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

[How to install Icarus Verilog + Gtkwave in Ubuntu Linux and test it - YouTube](#) . You can refer to this video if you are finding problems with the above procedure.

For mac users:

- To install iverilog:

```
brew install icarus-verilog
```

- To install GTKWave:

```
brew install gtkwave
```

For Windows Users:

- Install Icarus Verilog from the given link:- [Icarus Verilog for Windows \(bleyer.org\)](#)
 - Download the v-11 version (any subtype among the v-11 is fine).
 - Once downloaded , Run the file as administrator.
- Once you run the setup file
 - Accept the license agreement.
 - Select both the Components given (MinGw and GTKwave).
 - Remember the destination of the installed file (cause you will run the programs from there).
 - In the last step , select the check box which says “Add executable folder(s) to the user PATH.

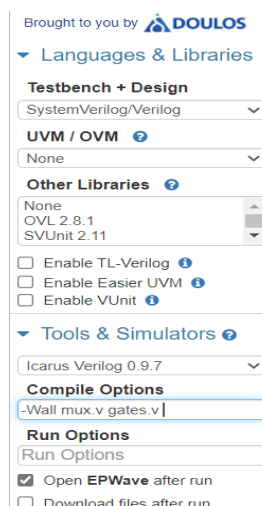
The above Setup comes with GTKWave so you won't be required to download it separately.

- Once Installed, you need to include the path in the environment Variable
 - Open the iverilog folder.
 - Copy the path of the bin folder.
 - Search for "Systems" in the windows search bar. Open the Systems.
 - Select Advanced System Setting. Click on Environment variables.
 - Click on New. Give any name to the Variable name (example: Iverilog) and paste the path of the bin folder path(which you have copied in earlier steps mentioned) in variable value.
 - [Icarus verilog + GTKWave installing and running | Free software for verilog HDL - YouTube](#). You can watch this video if you are having difficulty in following the above procedure.

EDA Playground ([Edit code - EDA Playground](#))

- You need to have internet access to run your verilog codes. This environment provides two places to write codes, one for modules and another for writing testbench.
 - To run the files, select these below given option in the left side toolbar available
 - Testbench+design :- System Verilog/Verilog
- Tools and Simulator :- Icarus Verilog 0.9.7
- Compile options :- give all your filenames with.v extension which you want to run (you can give multiple files here)

Below you can find the screenshot of the correct settings.



File Formats:

- Modules and testbenches must have the extension (.v).
- Dumpfiles should have the extension (.vcd)

Execute:

- To compile the written code, just type

```
iverilog <fileName> <names of any included modules>
```
- To execute the compiled code, as usual run

```
./a.out
```
- To check the waveforms on GTKWave, use the command

```
gtkwave <dumpfileName>
```
- To observe waveforms on GTKWave, select the module name in the left side section, then drag whichever variable you want to observe.
- The above Procedures need to be executed in the terminal (make sure you enter the terminal when you are in the specific folder).

-----End of Document-----