

# Commands to Run Verilog Open-Source Using iverilog

Anil Kumar Gundu

November 2025

## Icarus Verilog + GTKWave Workflow

The basic workflow to compile, simulate, and view waveforms for a Verilog module using **Icarus Verilog** and **GTKWave** is as follows:

### 1. Compile the Verilog code

Command:

```
iverilog -o <output_executable> <source_files>
```

**Explanation:**

- `iverilog` : Icarus Verilog compiler
- `-o <output_executable>` : Name of the compiled simulation executable
- `<source_files>` : List of Verilog files (module + testbench)

**Example:**

```
iverilog -o dff_tb dff_sync_reset.v tb_dff_sync_reset.v
```

Here, `dff_sync_reset.v` is the DFF module and `tb_dff_sync_reset.v` is the testbench. The output executable is `dff_tb`.

### 2. Run the simulation

Command:

```
vvp <output_executable>
```

**Explanation:**

- `vvp` : Runs the compiled simulation
- Generates a `.vcd` waveform file if `$dumpfile` and `$dumpvars` are used in the testbench

**Example:**

```
vvp dff_tb
```

### 3. View the waveform

Command:

```
gtkwave <vcd_file>
```

**Explanation:**

- `gtkwave` : Opens GTKWave waveform viewer
- `<vcd_file>` : Waveform file generated by simulation

**Example:**

```
gtkwave dff_sync_reset.vcd
```

## Full Example Workflow

```
# 1. Compile
iverilog -o dff_tb dff_sync_reset.v tb_dff_sync_reset.v

# 2. Run simulation
vvp dff_tb

# 3. View waveform
gtkwave dff_sync_reset.vcd
```