

Using the Icarus Verilog

1. Download the full Icarus package from:
<http://bleyer.org/icarus/>
2. It is recommended to download the latest stable release (currently: [iverilog-0.9.7_setup.exe \(latest stable release\)](#) [10.5MB])
3. Install the software, and make sure to select "full installation" and "Add Executables to Windows Path" option.
4. Open any command prompt, and type "iverilog", you should get a message saying something like "iverilog: no source files". At this point your installation is correct.
5. If you type "iverilog" and it is not recognized, then you have to set the PATH environment variable by yourself.
6. Control Panel > System and Security > System
7. Click on "Advanced System Settings" and select "Environment Variables"
8. Edit the "PATH" environment variable by appending ";c:\iverilog\bin" to it at its end.
9. Now everything should work fine.
10. Let's now play with the simulator. We will write a verilog code to generate a clock. Write the following code in an editor and name it: test.vl

```
module testbench(clk);  
output clk;  
reg clk;
```

```
initial  
begin  
    $dumpfile("test.vcd");  
    $dumpvars;  
    clk = 1'b1;  
    #50 $finish;  
end
```

```
always  
begin  
    #5 clk = ~clk;  
end  
endmodule
```

These are simulator directives to generate the waveform file "test.vcd"

11. Now open a command prompt from the folder where you saved your test.vl file. (File > Open command prompt).
12. Type `iverilog -o test test.vl`
13. This should compile the file and generate an output file called "test"
14. Now type: `vvp test`
15. This will generate a file called "test.vcd" that contains the waveform
16. Now open the waveform file by typing "gtkwave"
17. A new application will open. Go to the file menu and select "Open New Tab" and open your "test.vcd" file.
18. Click on the name of your module, which is in our example "testbench". Its variables will appear. Drag and drop the "clk" variable into the drawing part of the window, and you will see the waveform.