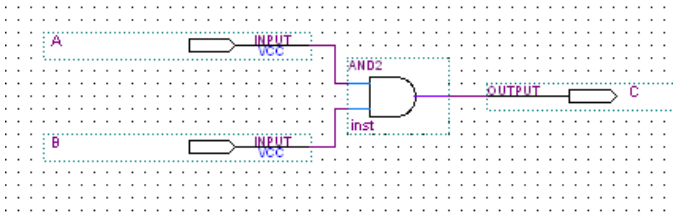
EECS 270 Quick Reference Simulation Guide



Design File (my_design.bdf)

```
`timescale 1 ns/1 ns //time scale for the test bench

module my_testbench_module(); // the test bench module

reg A_s, B_s; //define stimulus input ports
wire C_s; //define stimulus output port

//the following connects the test procedure below to my_design
my_design t1(.A(A_s), .B(B_s),.C( C_s));

//what follows is the test procedure
initial begin

//test all possible input conditions
//#10 means wait 10 time scale units (defined to be ns above)
A_s <= 0; B_s <= 0;  //set 1st test case
#10;                 //wait 10ns
A_s <= 1; B_s <= 0;  //set 2nd test case
#10;                 //wait 10ns
A_s <= 0; B_s <= 1;  //set 3rd test case
#10;                 //wait 10ns
A_s <= 1; B_s <= 1;  //set 4th test case
#10;                 //wait 10ns
end
endmodule
```

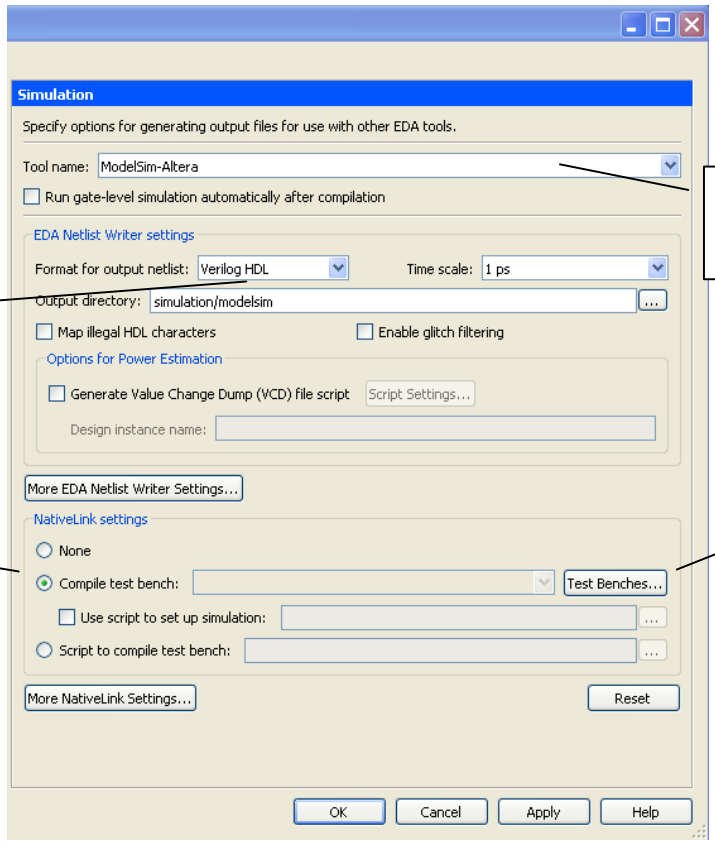
Test Bench File (my_testbench_file.v)

Test Bench Essentials

1. **The top level test bench module name “my_testbench_module” is arbitrary, but unique.**
2. **The design file name “my_design” must be named exactly as the design file name.**
3. **The instance name “t1” is arbitrary, but unique.**
4. **The port references (pins) to my_design must use the same name as the design (A, B and C).**
5. **The port (variable) references to the simulation must follow the declaration naming and be unique A_s, etc**
6. **The output simulation port “C” needs to be declared as a wire.**
7. **Naming should start with a lower or upper case letter.**

Setting Up the Quartus Project for Simulation

Go To: Assignments→Settings→Simulation and fill the field in accordingly.



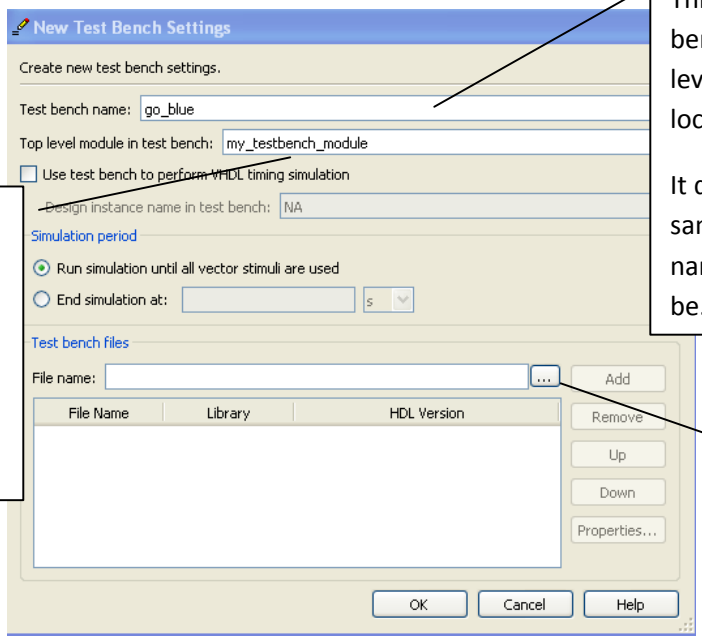
Select the ModelSim-Altera tool

Select Verilog

Select Compile test bench

Select Test Benches to link your test bench module name and file.

Go To: Test Benches and Select New



This field names the test bench and associates the top level module name and file location for the test bench.

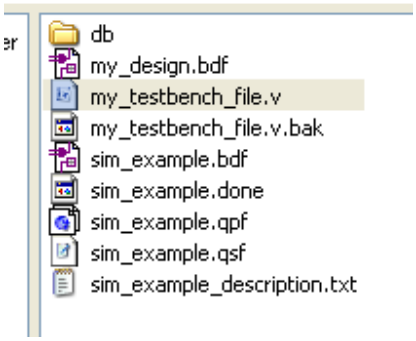
It does NOT have to be the same name as the module name or file name, but can be.

This field **MUST** be named the module name in your test bench file.

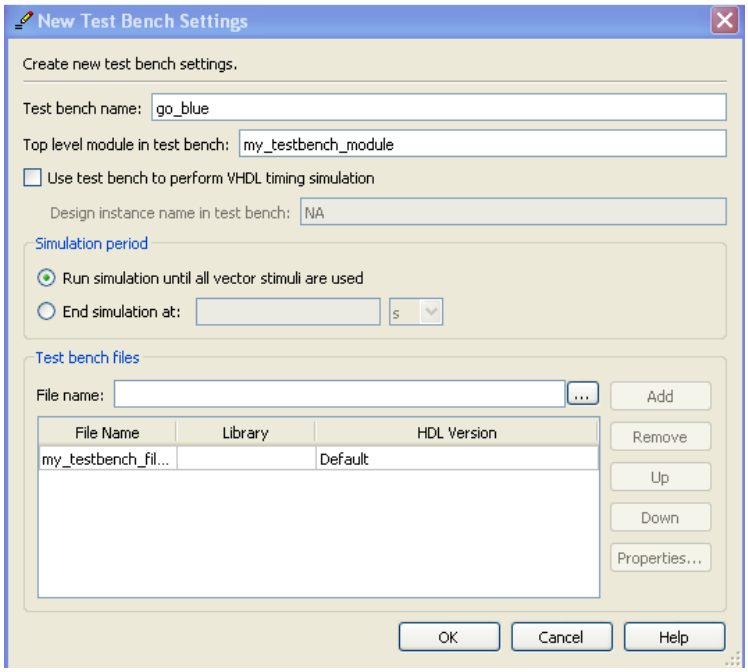
It does NOT have to be the same as the file name or test bench name field, but can be.

Browse to test bench file.

Browse, Select and Open my_testbench_file

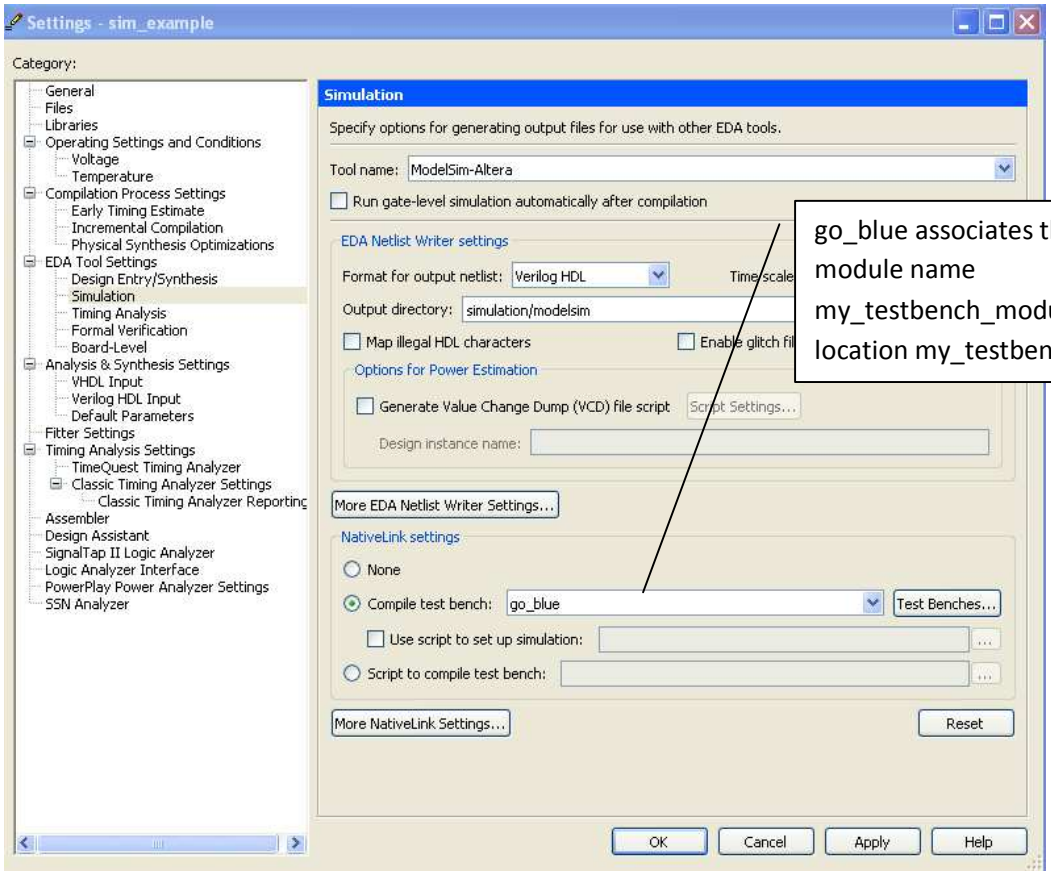


Add to test bench library



Acknowledge and return to the Simulation Settings Window

Note you must click APPLY for these setting to be accepted.

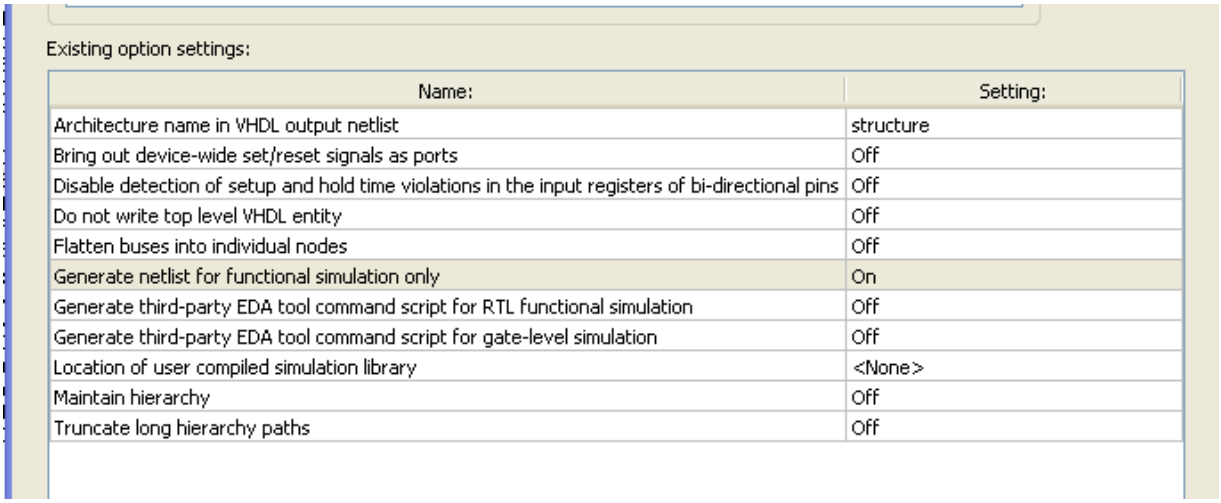


Selecting Timing or Functional Simulation

The default is for Timing Simulation

To Select Functional Simulation, Select "More ED Netlist Writer Settings"

Select On for "Generate netlist for functional simulation only"



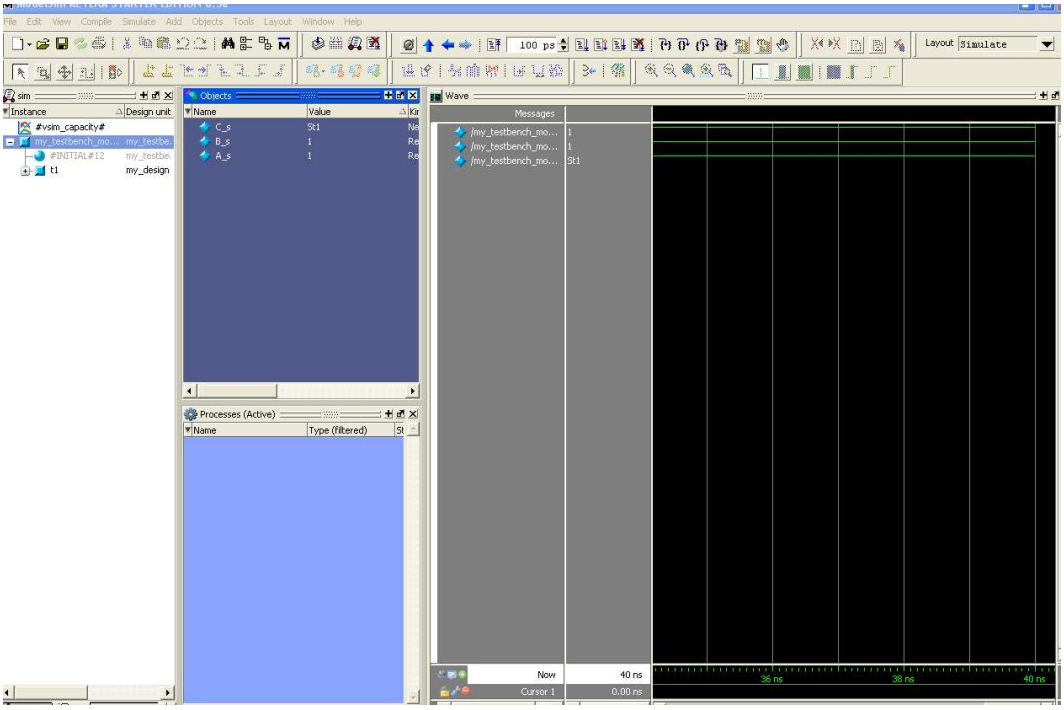
Existing option settings:

Name:	Setting:
Architecture name in VHDL output netlist	structure
Bring out device-wide set/reset signals as ports	Off
Disable detection of setup and hold time violations in the input registers of bi-directional pins	Off
Do not write top level VHDL entity	Off
Flatten buses into individual nodes	Off
Generate netlist for functional simulation only	On
Generate third-party EDA tool command script for RTL functional simulation	Off
Generate third-party EDA tool command script for gate-level simulation	Off
Location of user compiled simulation library	<None>
Maintain hierarchy	Off
Truncate long hierarchy paths	Off

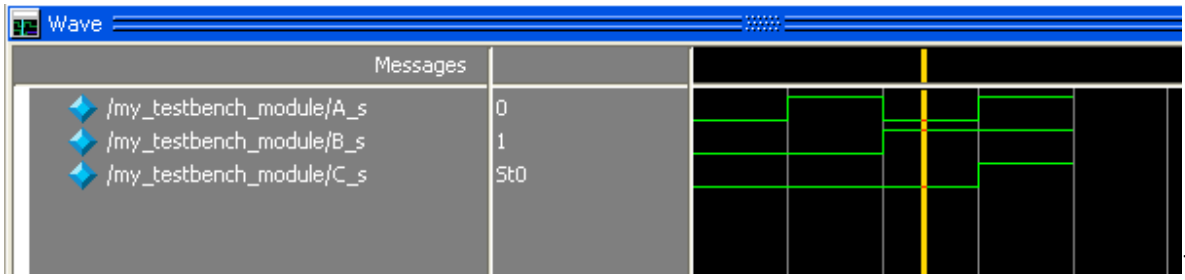
To Run Simulation

1. **Make Sure all Settings Were Saved**
2. **Processing→Start Compilation**
3. **Run EDA Simulation Tools→EDA Gate Level Simulation**

The following page should open.



Click in the waveform window to activate the cursor, right click and select zoom out or use the minus key. You should be able to manipulate the waveform screen to see the simulation results.



To add a second cursor, click on the plus sign in the lower left waveform window in the signal column. A second cursor will appear. The simulation time point and difference between the 2 cursors is displayed below on the time line.