

Notes for Using ModelSim-Intel

Robert Li

1. Requirements to simulate a Quartus project:

- 1) Before simulation, the Quartus project needs to be successfully compiled.
- 2) Change Quartus settings for EDA tools to use ModelSim-Intel (See Section 2).
- 3) Create a new input waveform file, or open an existing input waveform file (See Section 3).
- 4) In the “Simulation Waveform Editor” window, make sure the settings are correct. Then click “Simulation | Run Functional Simulation”. (Section 4)

PLEASE NOTE: *Starting from version 17.0, The Quartus gate-level timing simulation is NOT supported for Cyclone V, which is used on DE1-SoC boards. When simulate a design using ModelSim-Intel, timing simulation will give the same simulation result as the functional simulation.*

2. Quartus EDA tools settings for doing simulation using ModelSim-Intel :

- 1) In Quartus Prime, go to Tools | Options | ..., then select EDA Tools Options. At the right side of the window, set the correct path for “ModelSim”.

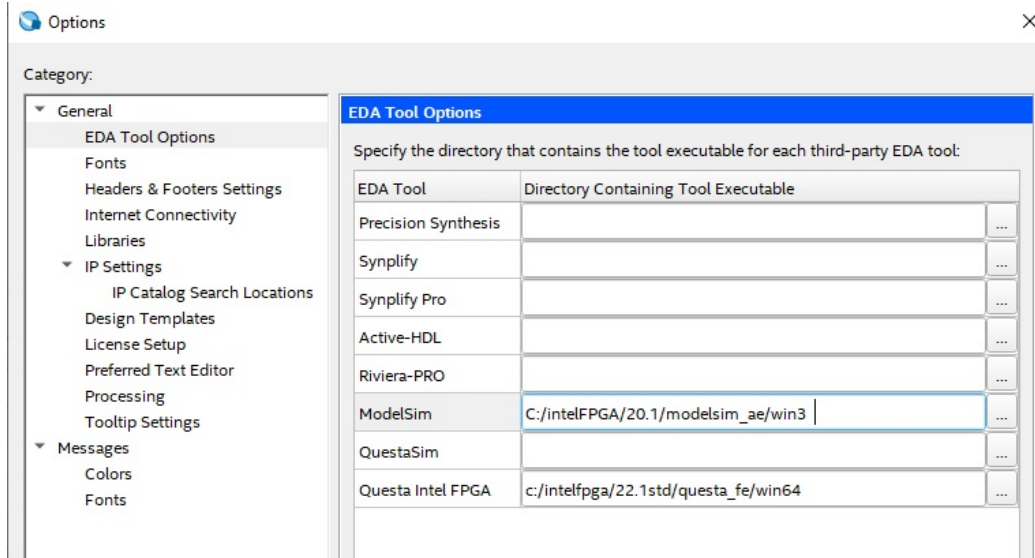
Depending on where the Modelsim_Intel is installed, in our case if you have followed our installation notes, the path to the ModelSim-Intel may look like:

“C:\intelFPGA\20.1\modelsim_ae\win32aloem”. Or like:

“C:\intelFPGA\20.1\modelsim_ase\win32aloem”

The folder “modelsim_ase” is for Modelsim Starter Edition (Altera Starter Edition). The folder “modelsim_ae” is for Modelsim Standard Edition (Altera Edition).

The ModelSim Starter Edition is free and does not require a license. Since our department has a license for ModelSim-Intel Standard Edition, at this step, set path to either folder will work for your lab projects.



- 2) In Quartus Prime, click “Assignment | Settings...”. From the “Category” window, select “Simulation” under the “EDA Tool Settings” category. In the right side panel, select and set the “Tool Name” to “ ModelSim”.

Leave other fields as default. Then click “Apply” and “OK” to close the setting window.

3. Create a new input waveform file and start simulation.

(For details of how to create input waveform files for ModelSim, please see the relative sections in “Introduction to Simulation of Verilog Designs”)

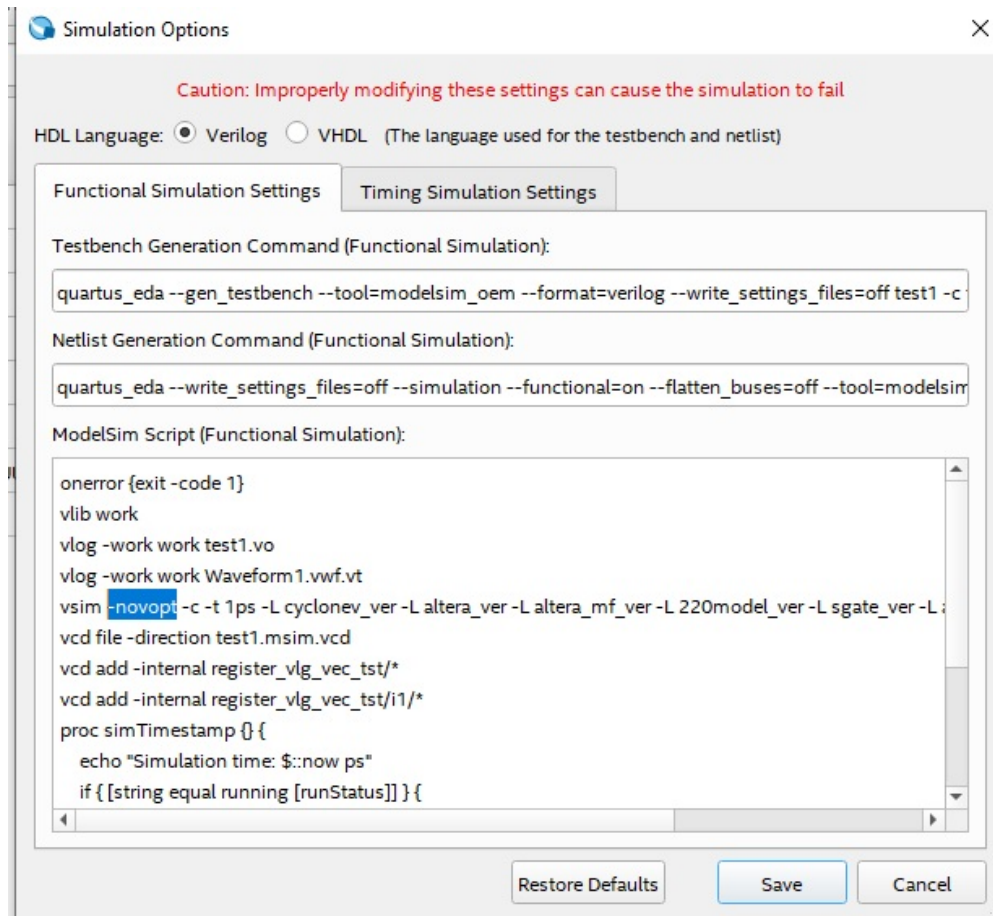
- 1) In Quartus, click File | New..., then select file type “University Program VWF”. This will open the “Simulation Waveform Editor”.
- 2) In “Simulation Waveform Editor”, click Edit | Insert | Insert Node or Bus
- 3) In the opened “Insert Node or Bus” window, click “Node Finder...”
- 4) In the “Node Finder” window, choose the appropriate Filter, then click “List”.
- 5) Select the desired Nodes. Then click “OK”. Then click “OK” again to insert the node or bus.
- 6) Specify the values at different time for the input signals.
- 7) Save the input waveform file.
- 8) Start simulation by clicking the “Run Functional Simulation” button on the tool bar in the Waveform Editor window, or by clicking menu Simulation | Run Functional Simulation.
However, since the version of the ModelSim is older than Quartus Prime, we need to

change a setting for simulation before we can run the simulation successfully for the first time. Please see the next section.

4. Change Simulation Settings for Simulation Waveform Editor:

After a waveform input file is created, you need to remove the option “-novopt” on one command line in “ModelSim Script” to make the simulation work. If your waveform input file is not saved using the default name “Waveform.wvf” you need to restore simulation settings (and remove the above mentioned option “-novopt”).

- 1) Remove the option “-novopt” on vsim’s command line:
 - I. In “Simulation Waveform Editor” window, click “Simulation | Simulation Settings”.
 - II. Select tab “Functional Simulation Settings” (as we need to, or can only, do functional simulation for DE1-SoC).
 - III. In the “Modelsim Script(Functional Simulation)” box, on the fifth line: “ vsim -novopt -c ...”, remove the option “-novopt”, because it is deprecated and will stop simulation. (See the figure below)
 - IV. Click Save to close the Simulation Option Window to save the new settings. Then start Simulation.



2) Fix the problem caused by the name of the waveform input file:

By default, the ModelSim expects the waveform input file is named as "Waveform.vwf". If you saved your waveform input file in other names, the system will complain and the simulation cannot start. The simplest solution to fix this problem is to restore the simulation settings using the "Restore Default" button.

- I. In "Simulation Waveform Editor" window, click "Simulation | Simulation Settings".
- II. Select tab "Functional Simulation Settings"
- III. Click button "Restore Defaults" at the bottom of the window.
However, restoring to default will restore the "-novopt" option for vsim command. You need to remove this option to make simulation work.
- IV. Click Save to close the Simulation Option Window to save the new settings. Then start Simulation.

To make things simple to remember: Before click Run Function Simulation, in the “Simulation Waveform Editor” window, click “Simulation | Simulation Settings”, then on the Functional Simulation tab, click the button “Restore Default” and then remove option “-novopt”.