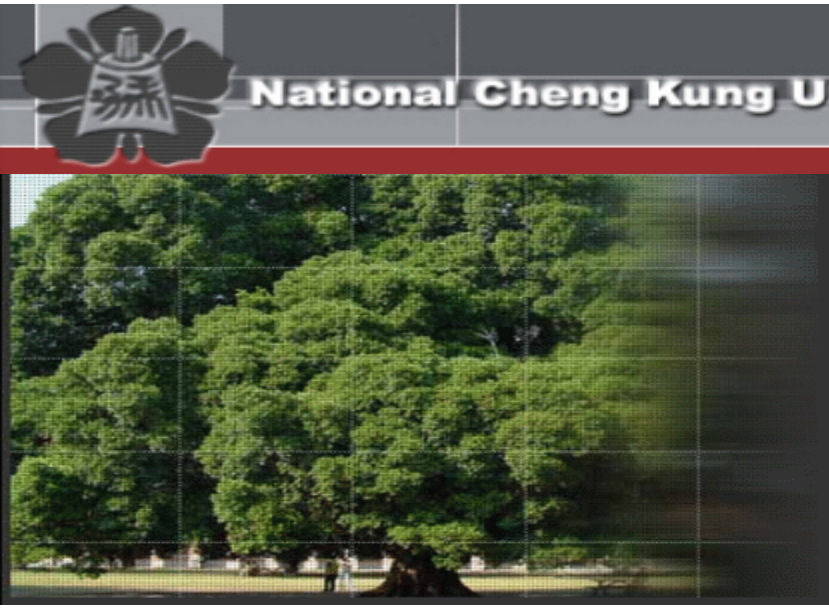
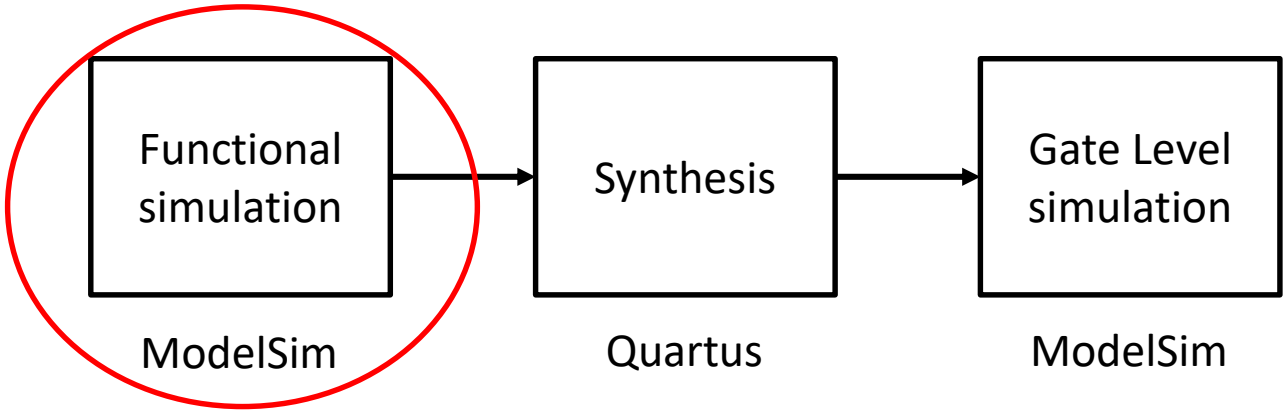


數位IC設計 Tool安裝&作業模擬教學

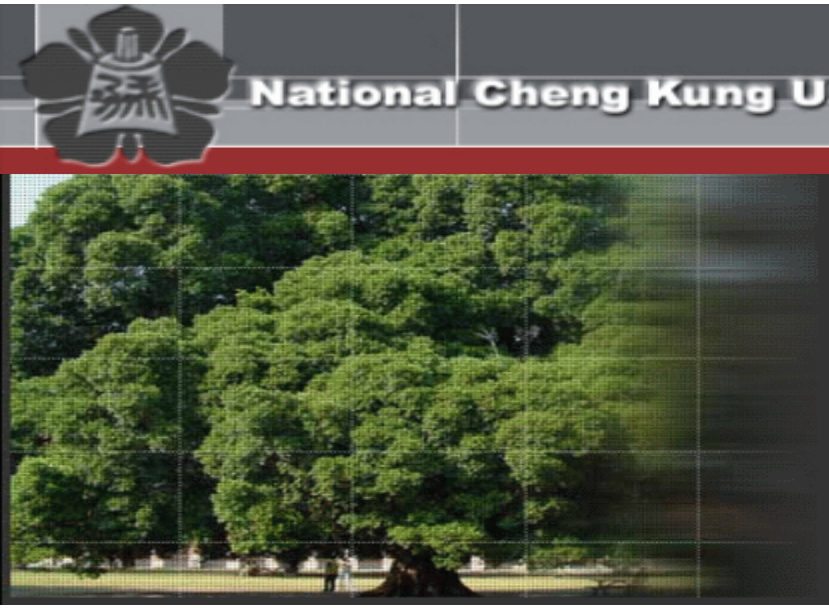
NCKU CSIE DICLAB

The simulation process

- The simulation process and the necessary tools in this class



- The tools can be downloaded from :
  - https://www.intel.com/content/www/us/en/software/programmable/quartus-prime/download.html#



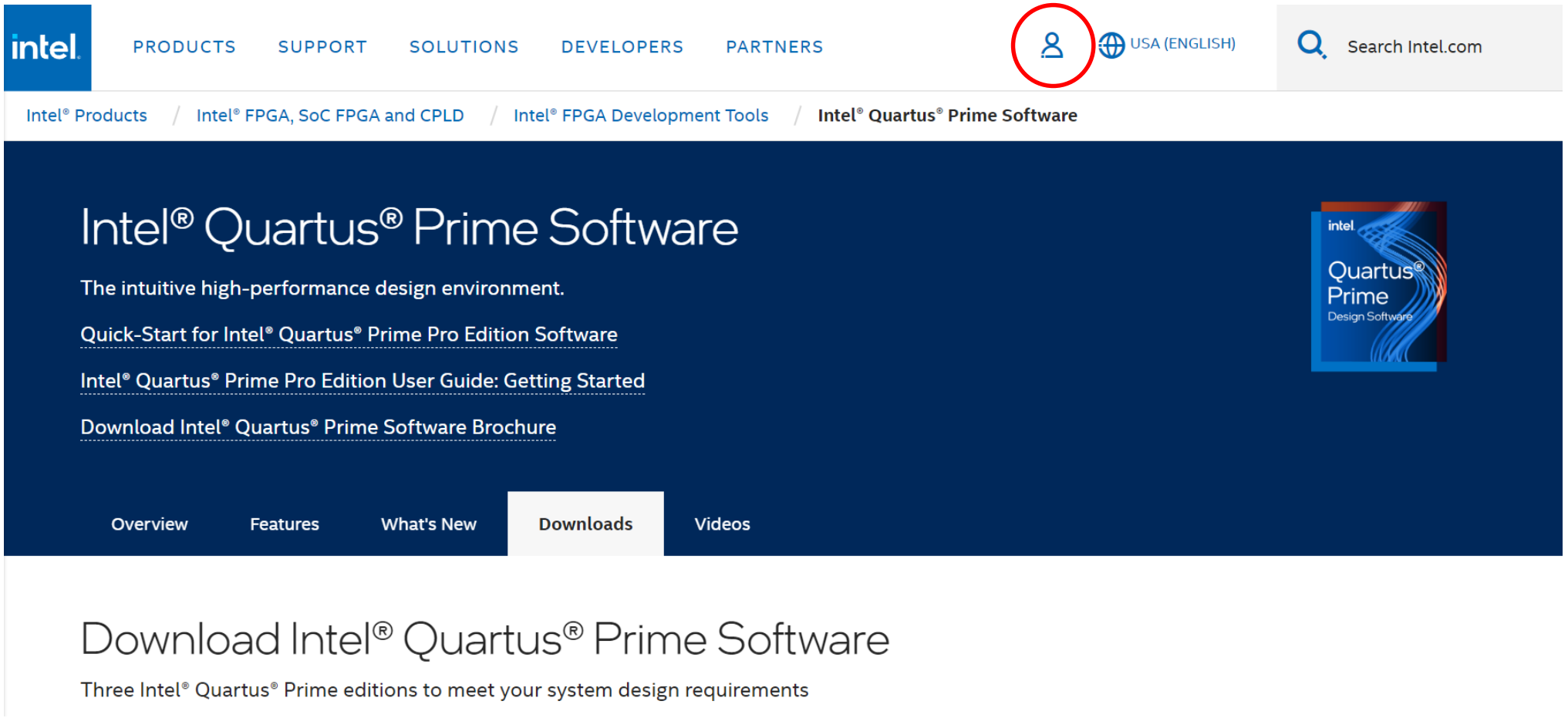
National Cheng Kung University

Download and install Modelsim

Download and install Modelsim

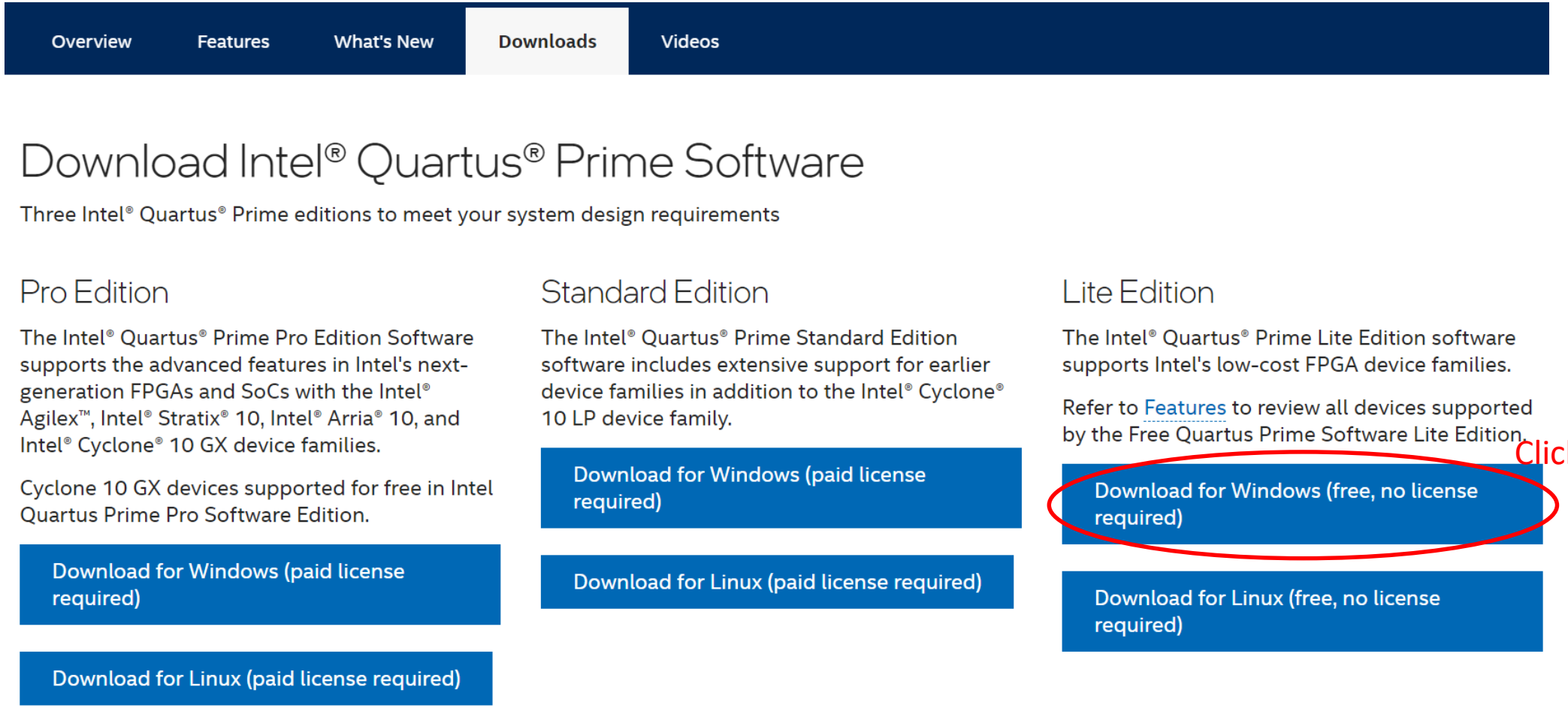
- Step 1: Register and sign in

Click here to create an account and sign in



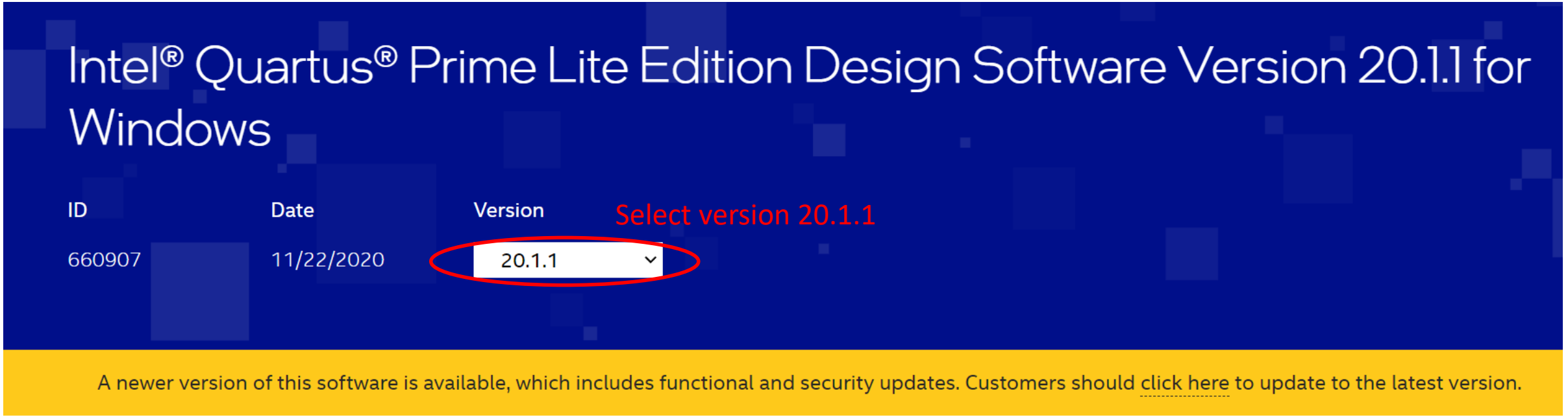
Download and install Modelsim

- Step 2: Click “ Download for Windows(free, no license required)“



Download and install Modelsim

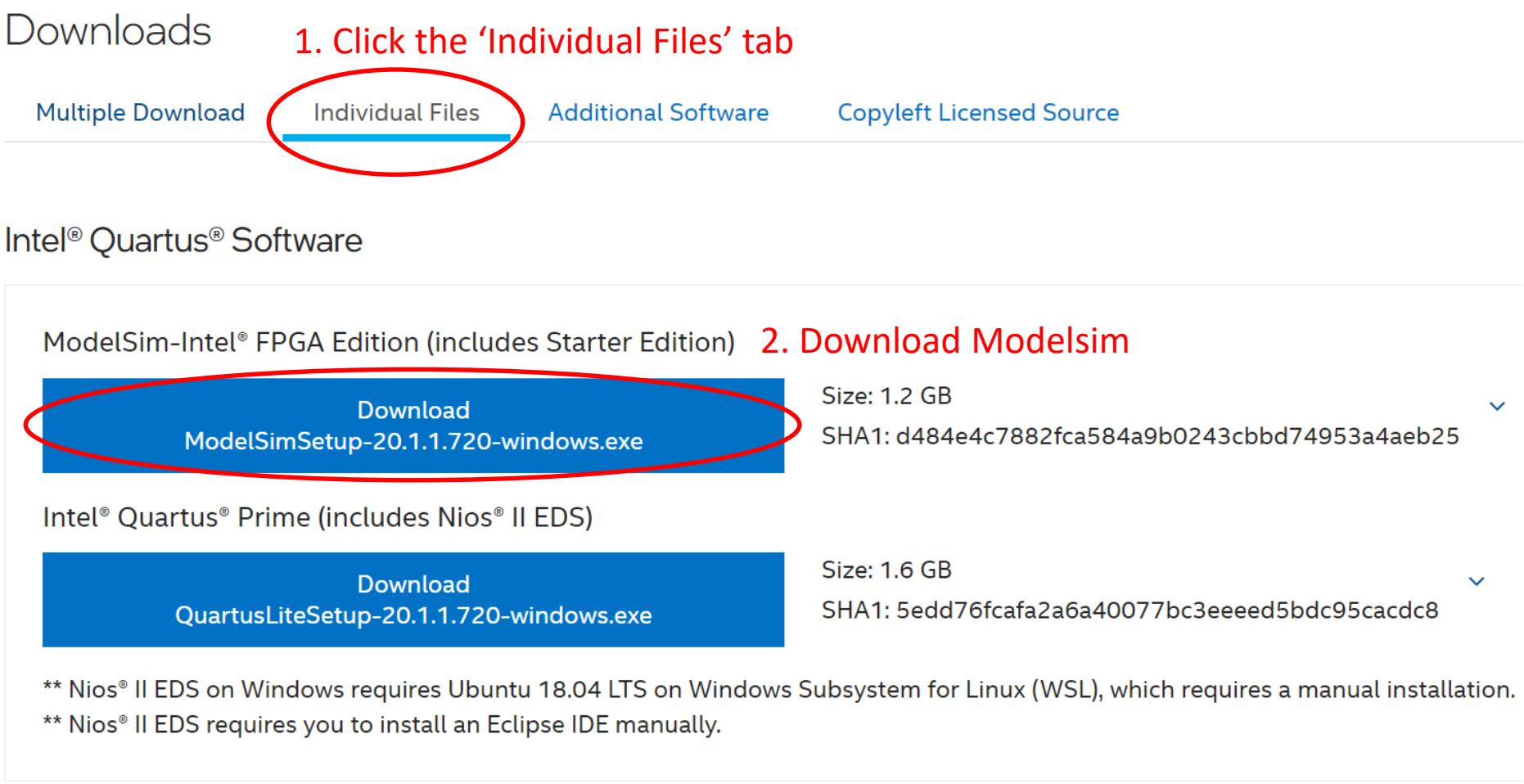
- Step 3: Select software version (20.1.1)



Users should upgrade to the latest version of the Intel® Quartus® Prime Design Software. The selected version does not include the latest functional and security updates. If you must use this version of software, follow the technical recommendations to help improve security. For critical support requests, please contact our support team.

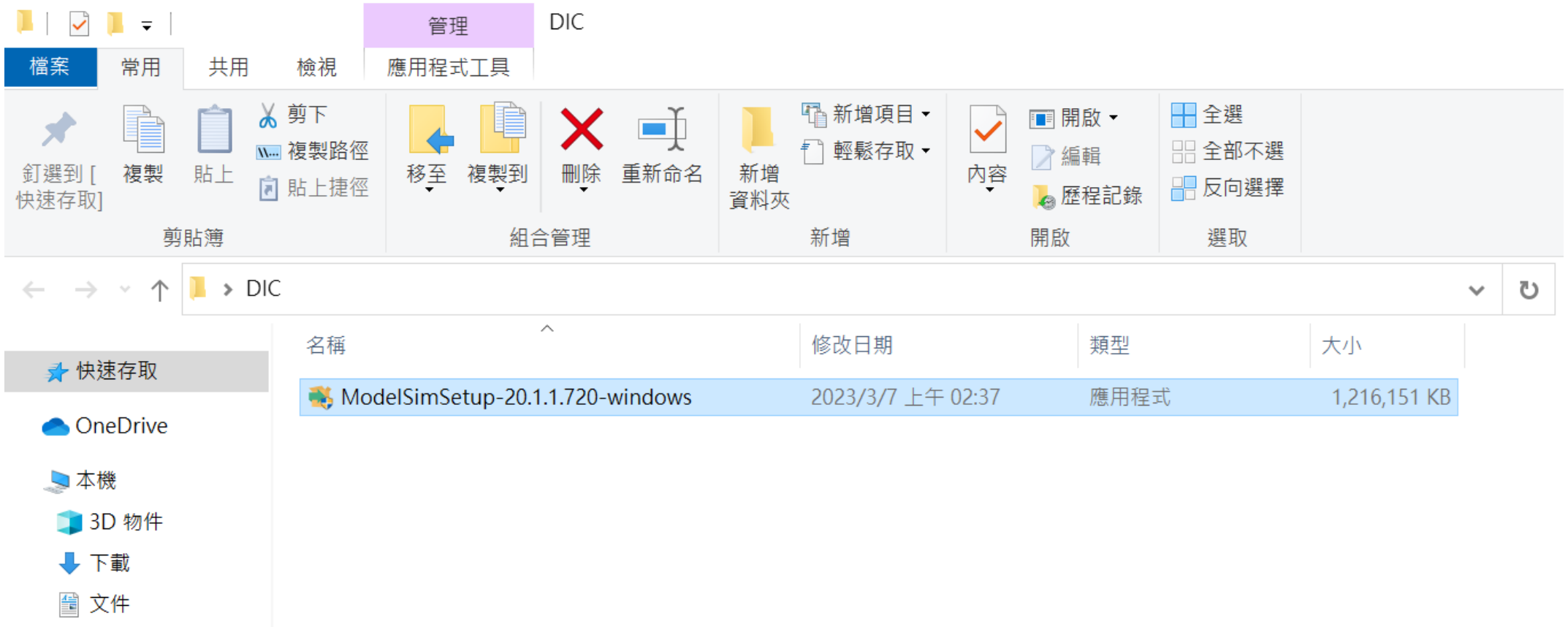
Download and install Modelsim

- Step 4: Download Modelsim setup execution file



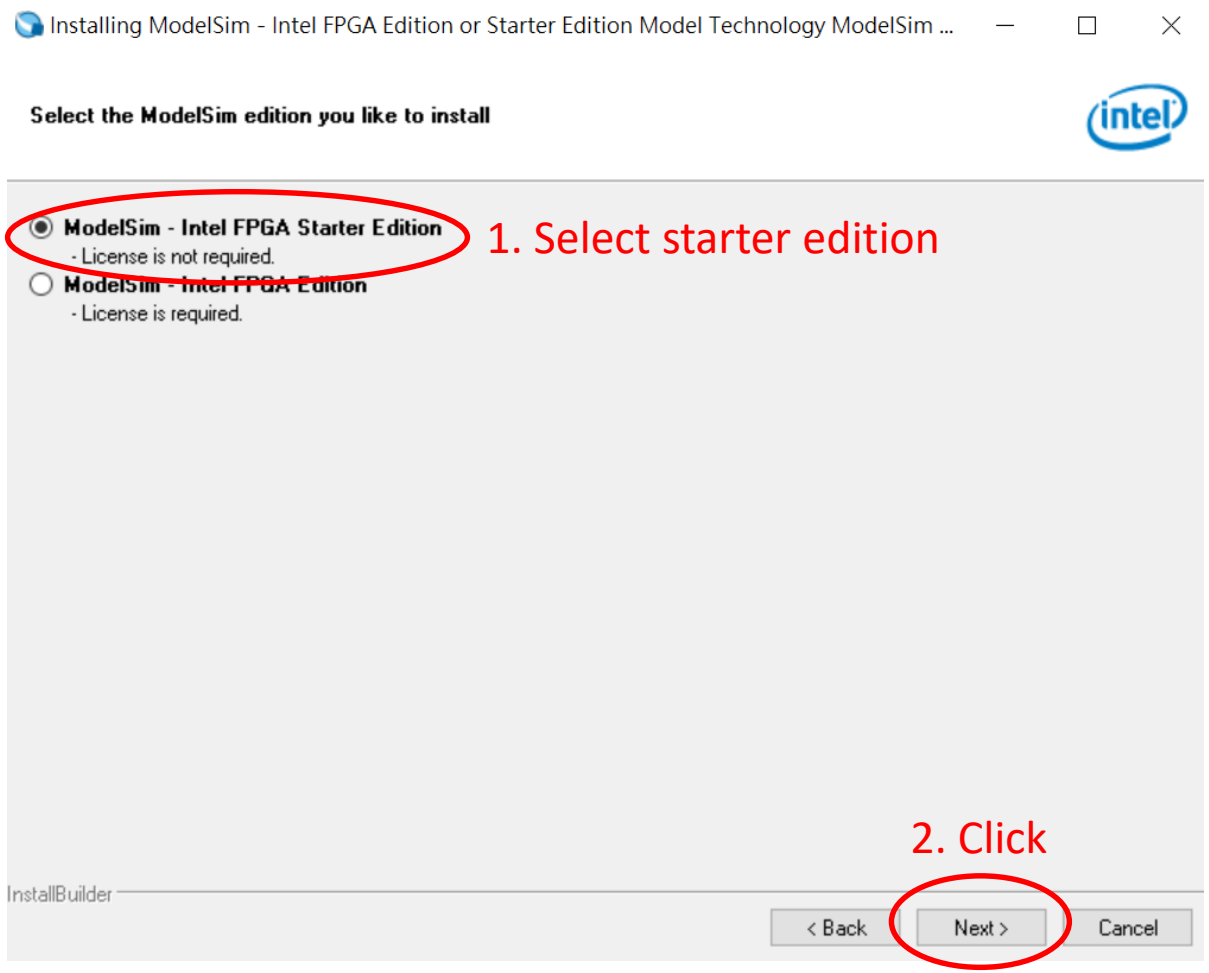
Download and install Modelsim

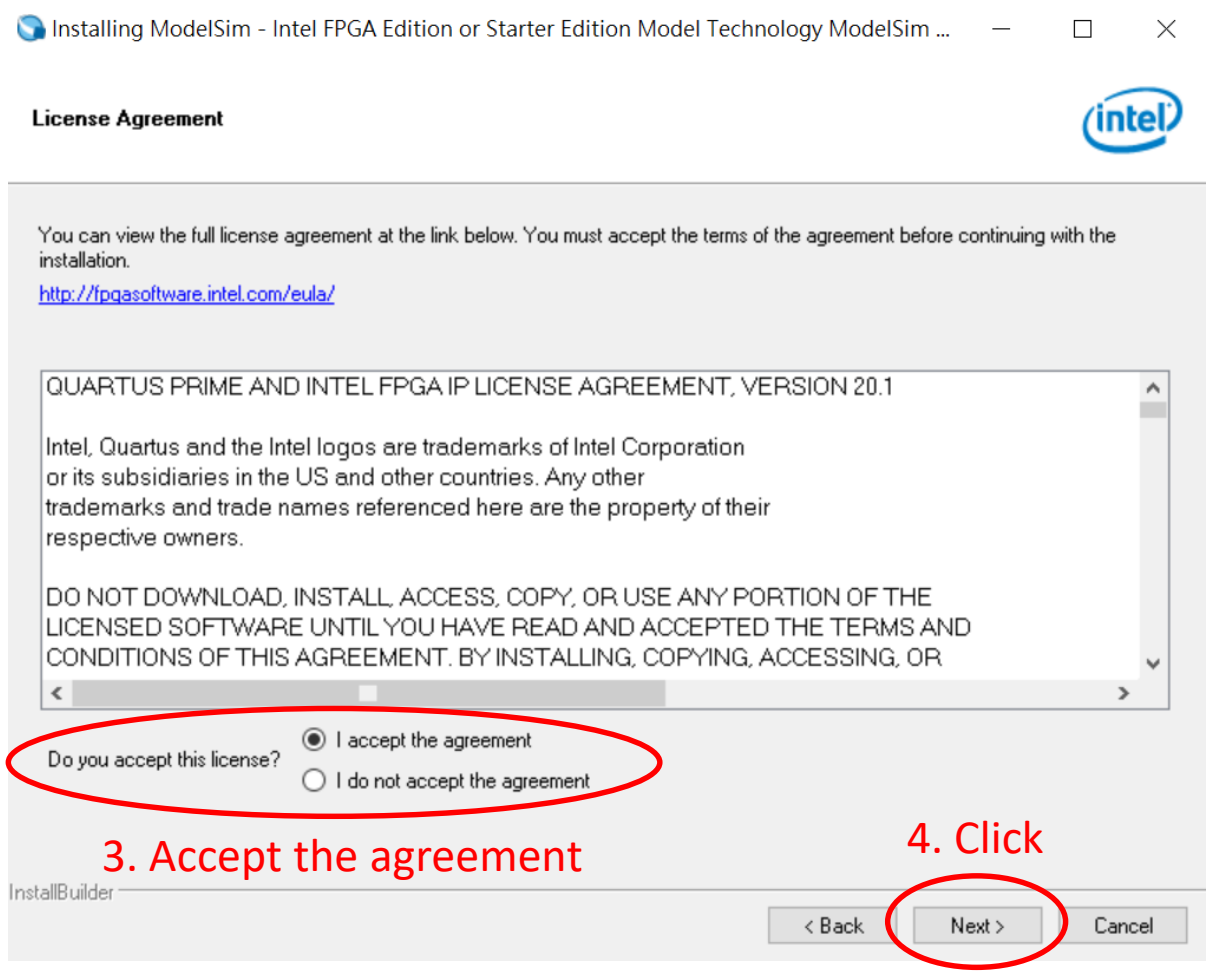
- Step 5: Installation



Download and install Modelsim

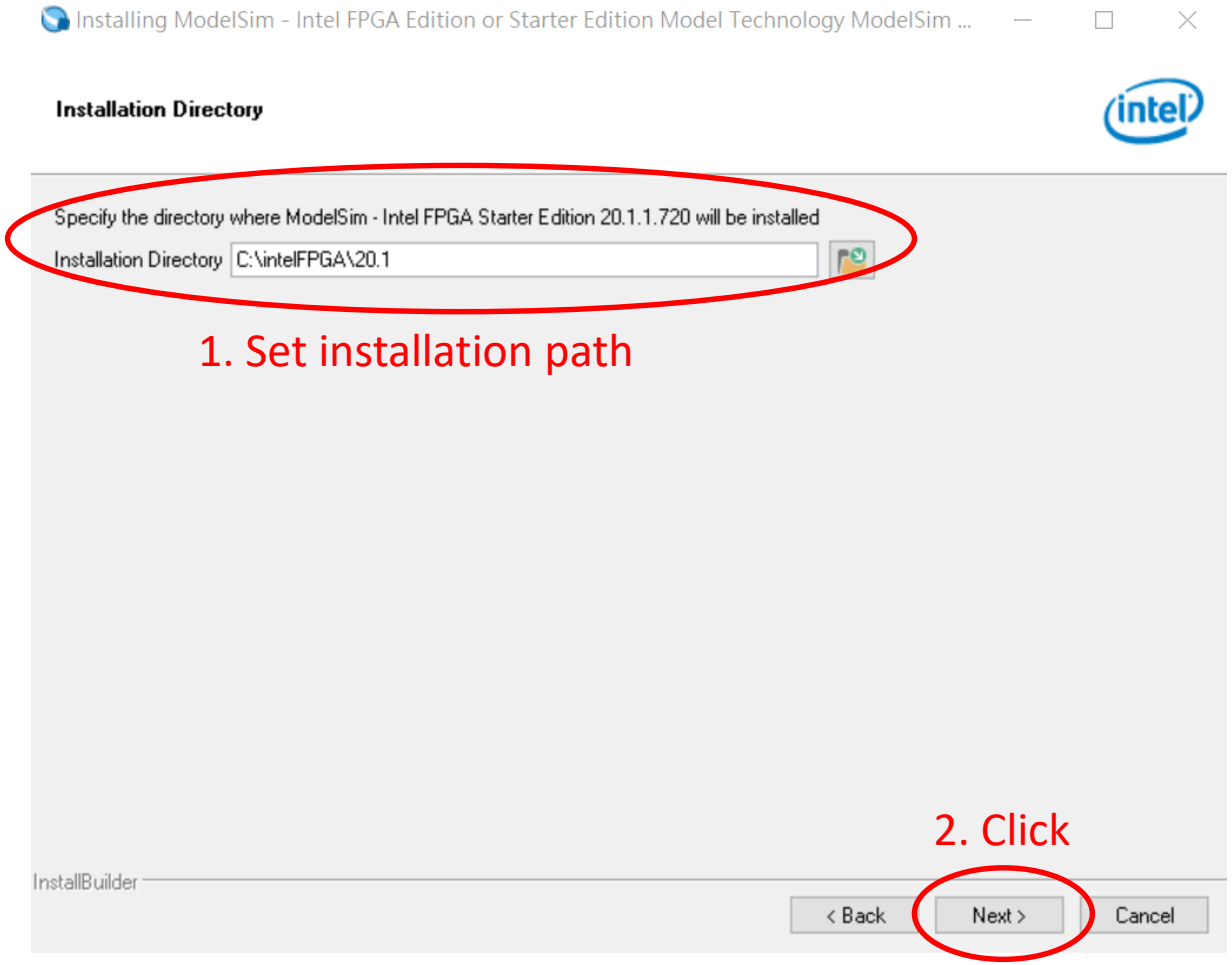
Step 5: Installation

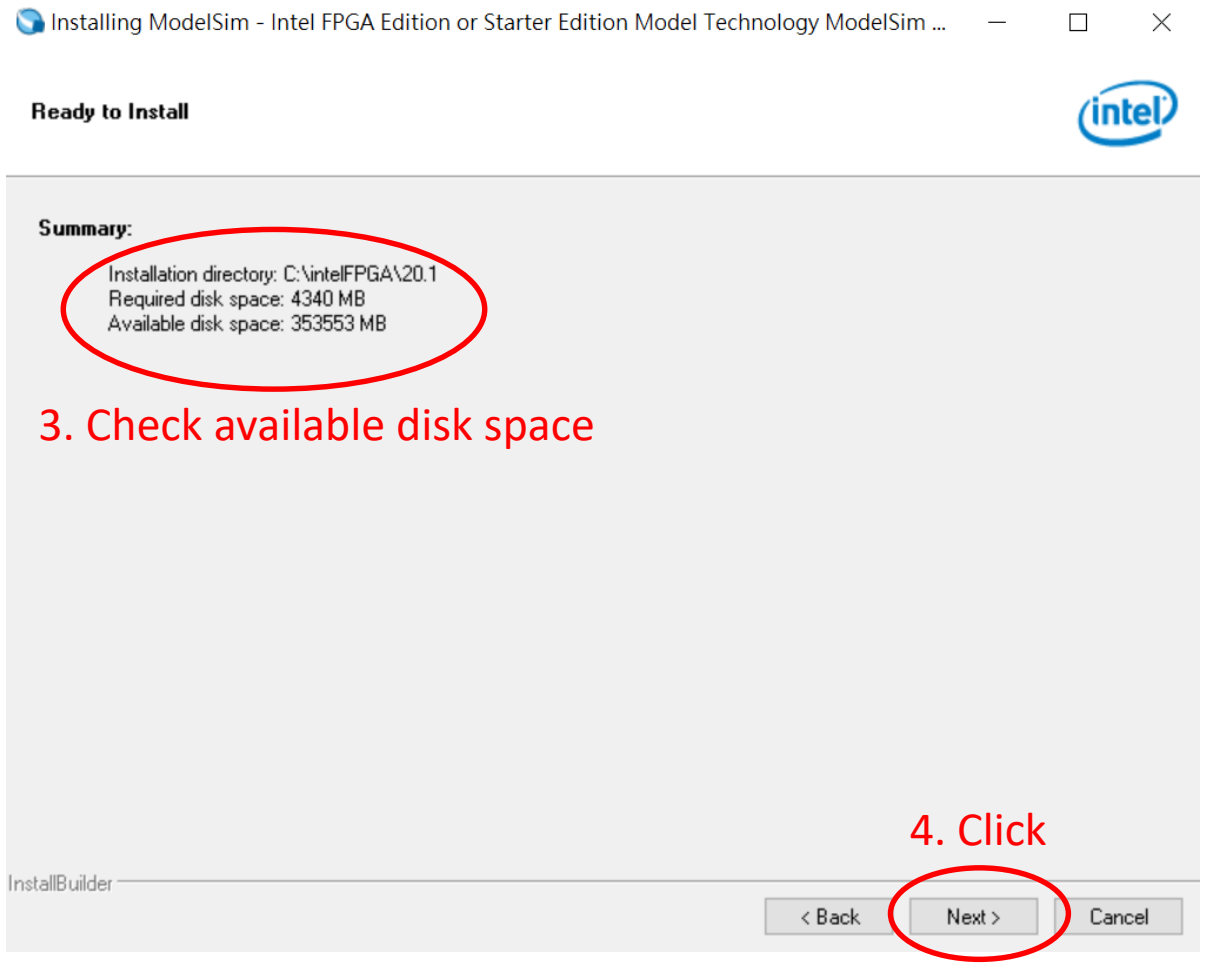




Download and install Modelsim

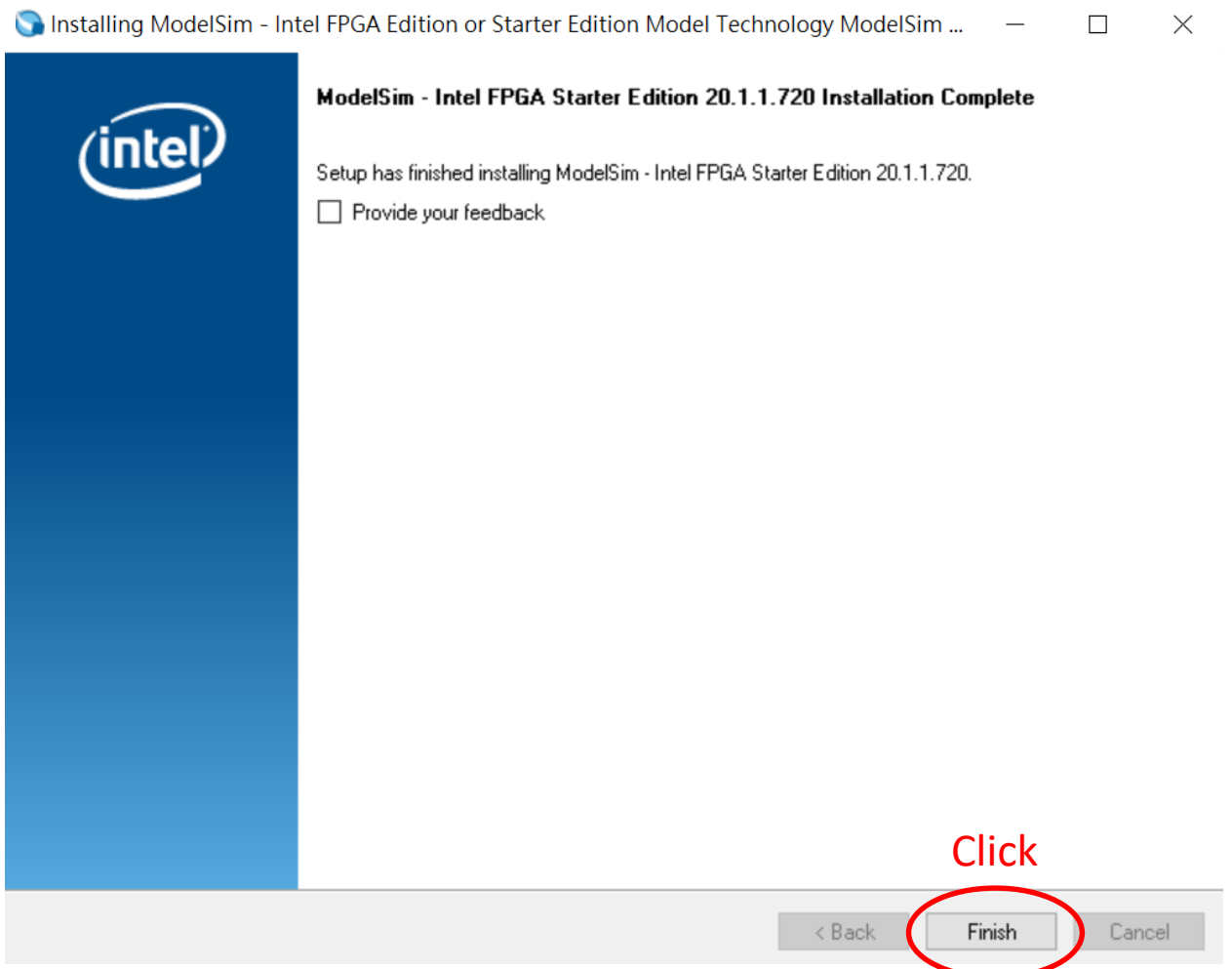
- Step 5: Installation

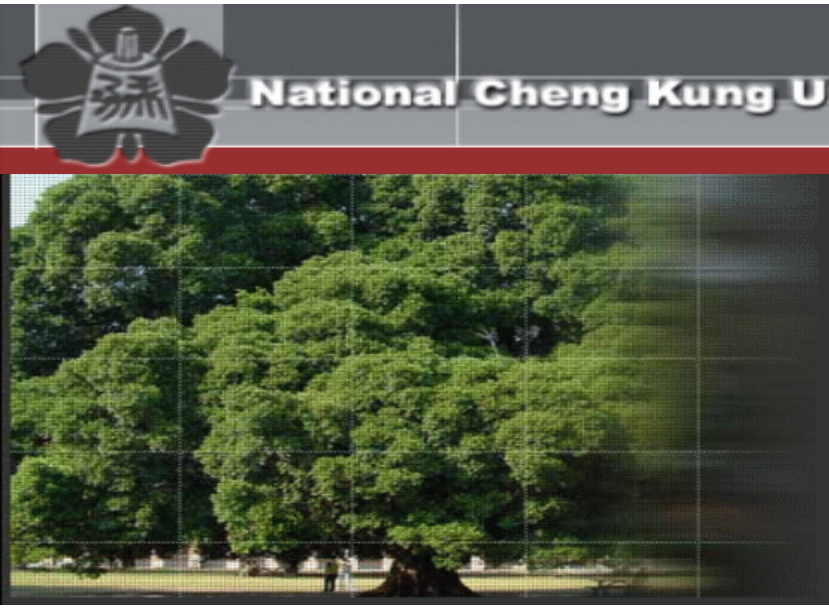




Download and install Modelsim

- Step 5: Installation



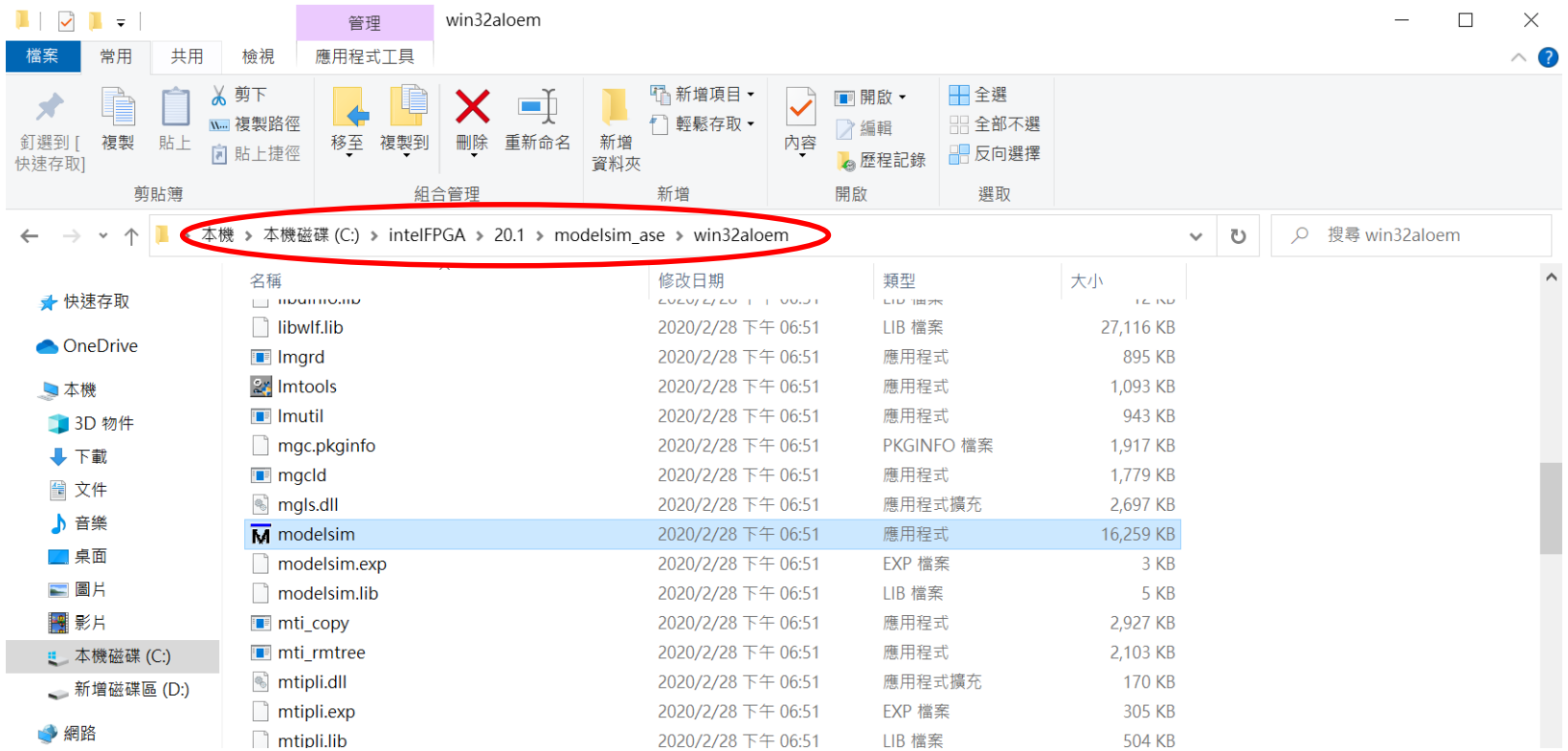


National Cheng Kung University

Functional Simulation by ModelSim

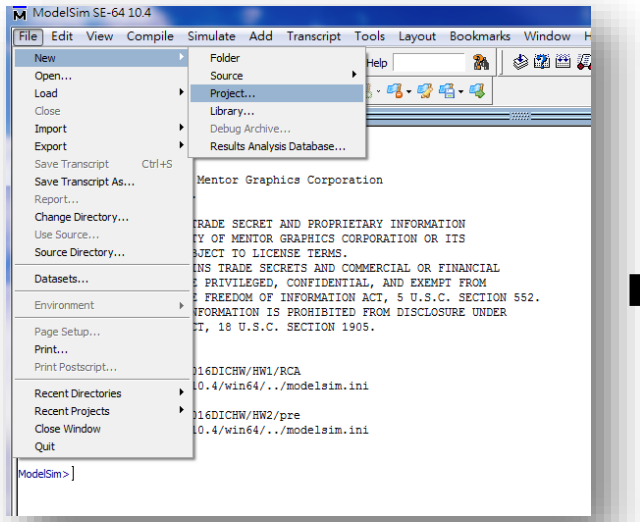
Functional Simulation by ModelSim

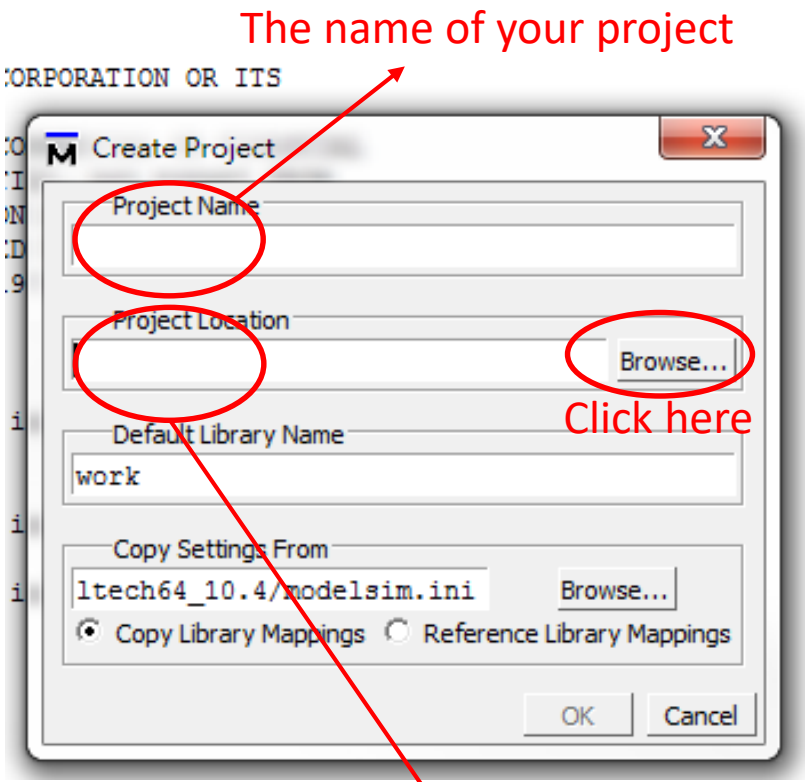
- Step 1: Run modelsim.exe
  - Your installation path
    - Ex: C:\intelFPGA\20.1\modelsim_ase\win32aloem\modelsim.exe

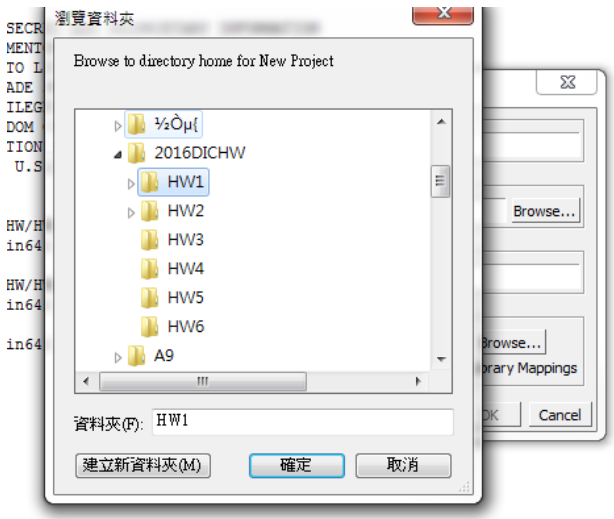


Functional Simulation by ModelSim

- Step 2: Create a new project
  - File -> New -> Project
  - Click “Browse…”



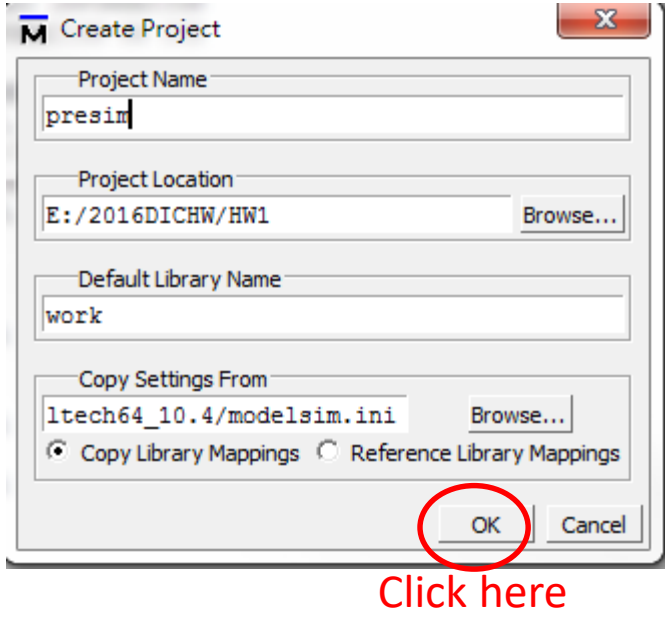


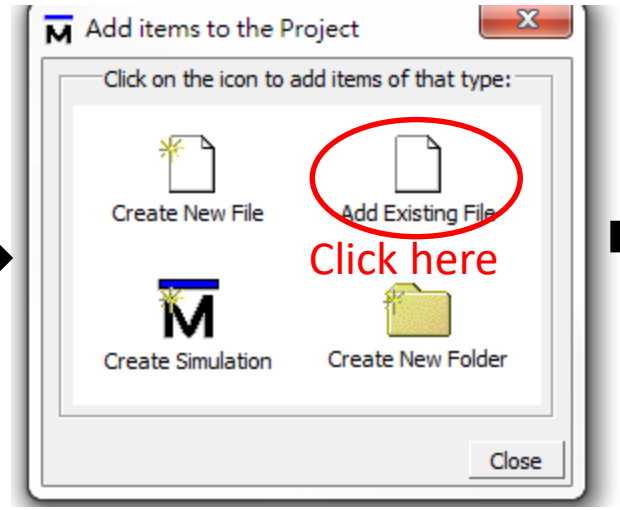


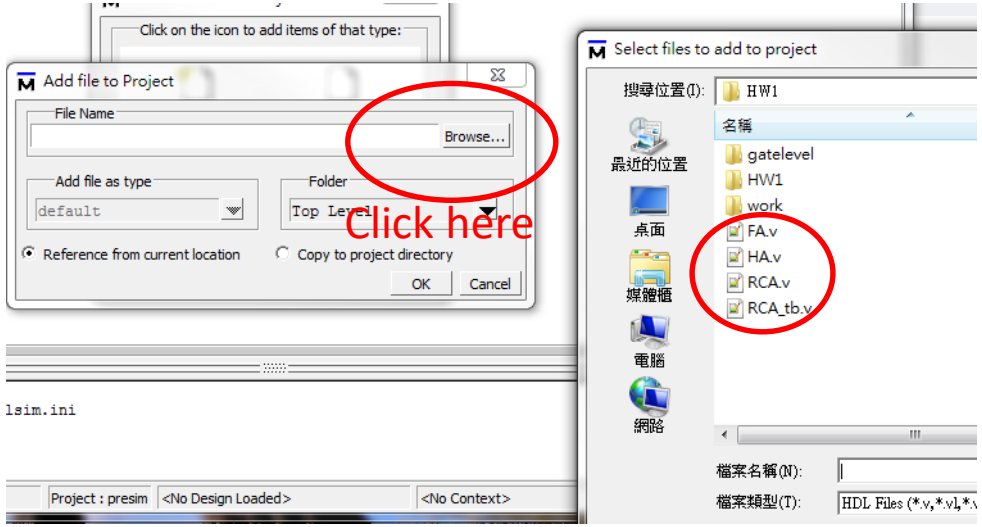
The file path should be ALL ENGLISH

Functional Simulation by ModelSim

- Step 2: Add the files
  - Select all the .v files including the testbench
    - Verilog code could be stored in any location , “add existing file” would refer to the file location automatically .
    - Remember to put the test data and golden data at the same directory of the testbench file.

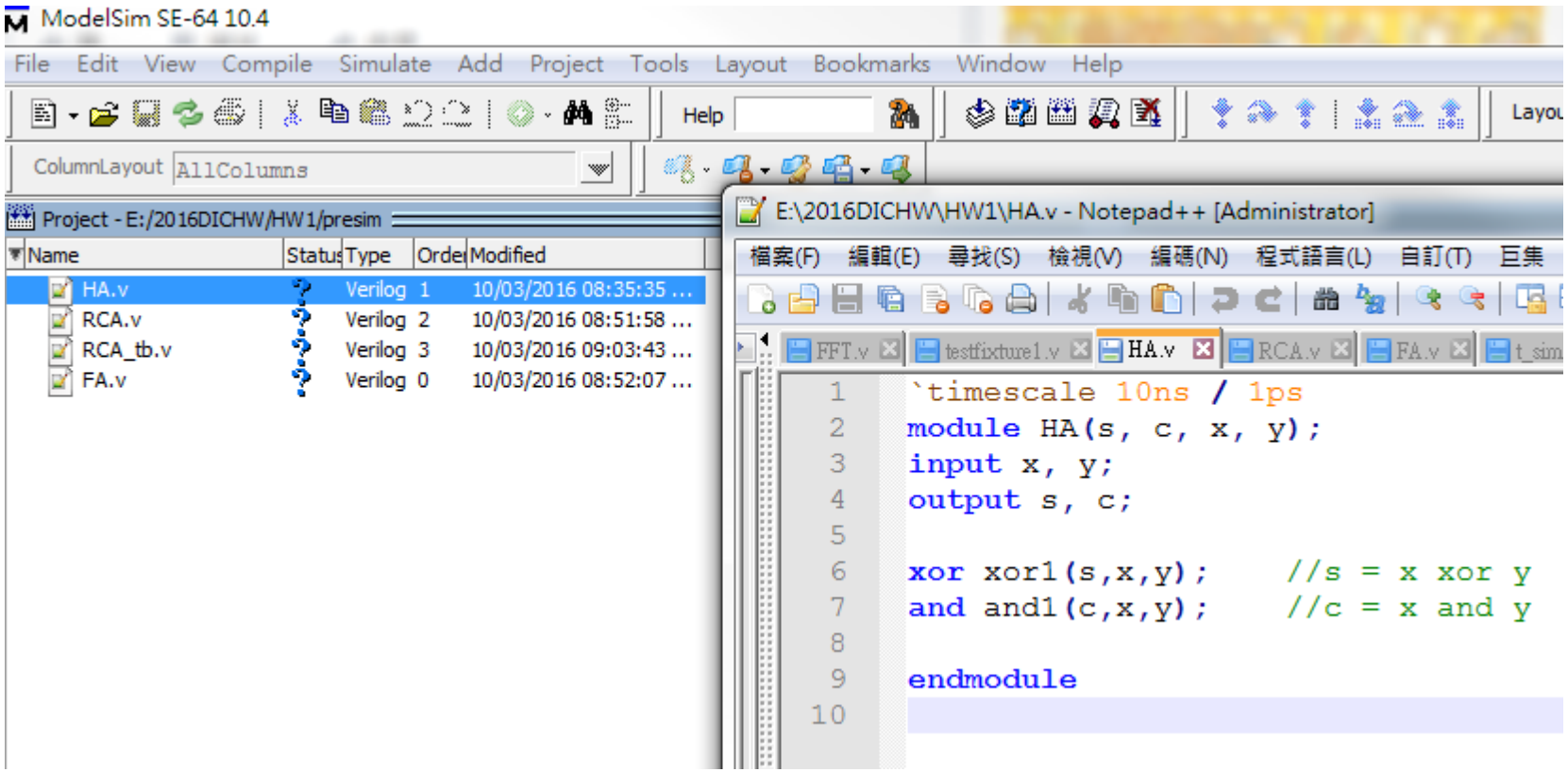






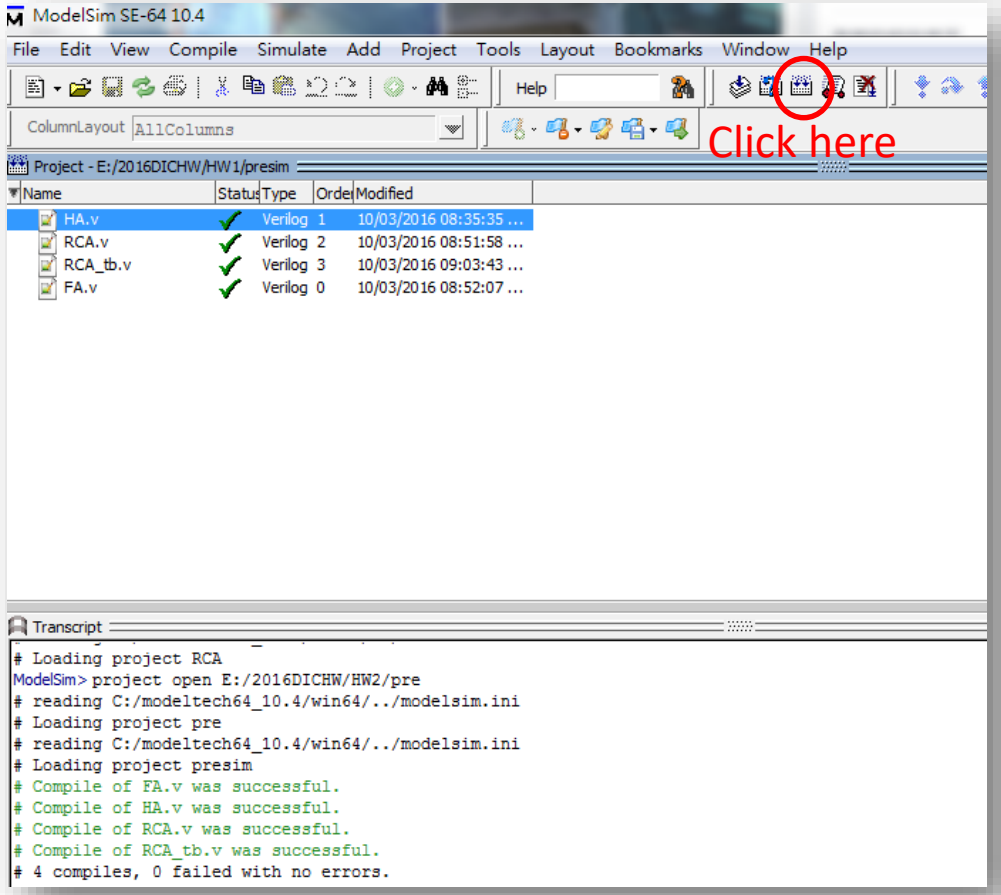
Functional Simulation by ModelSim

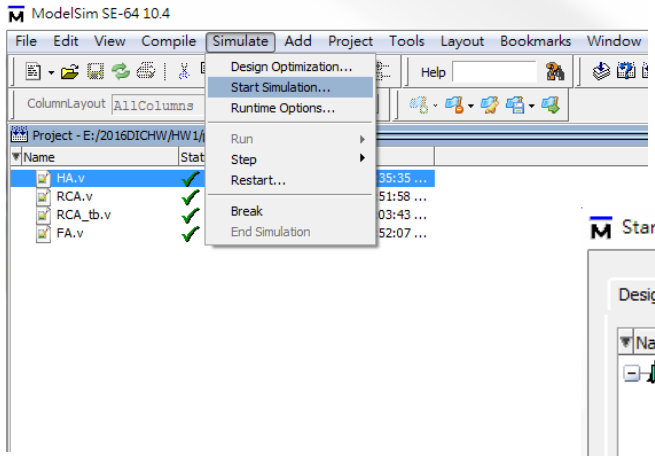
- Step 3 : Start coding…

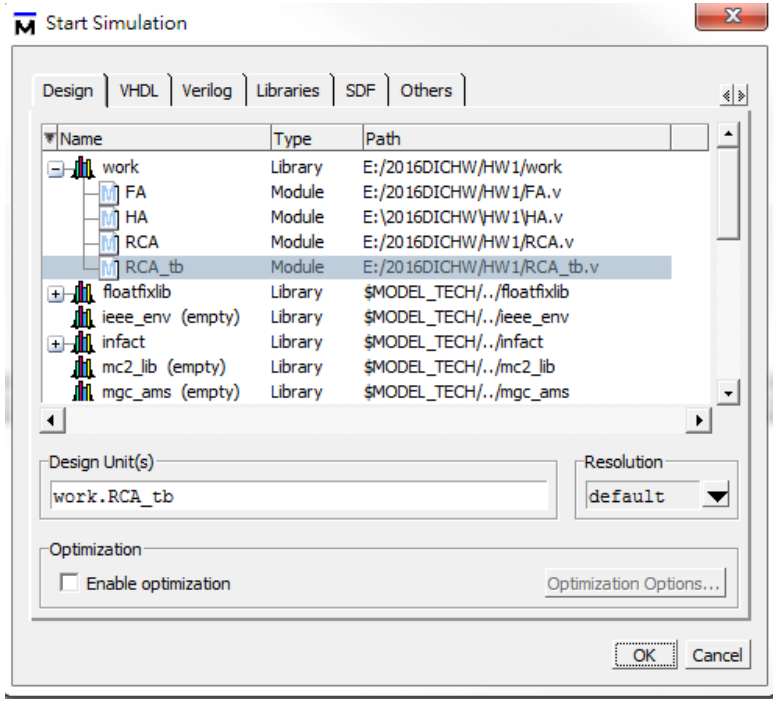


Functional Simulation by ModelSim

- Step 4: Compile & Start simulation
  - Compile -> Simulate -> Start Simulation->work->testbench file

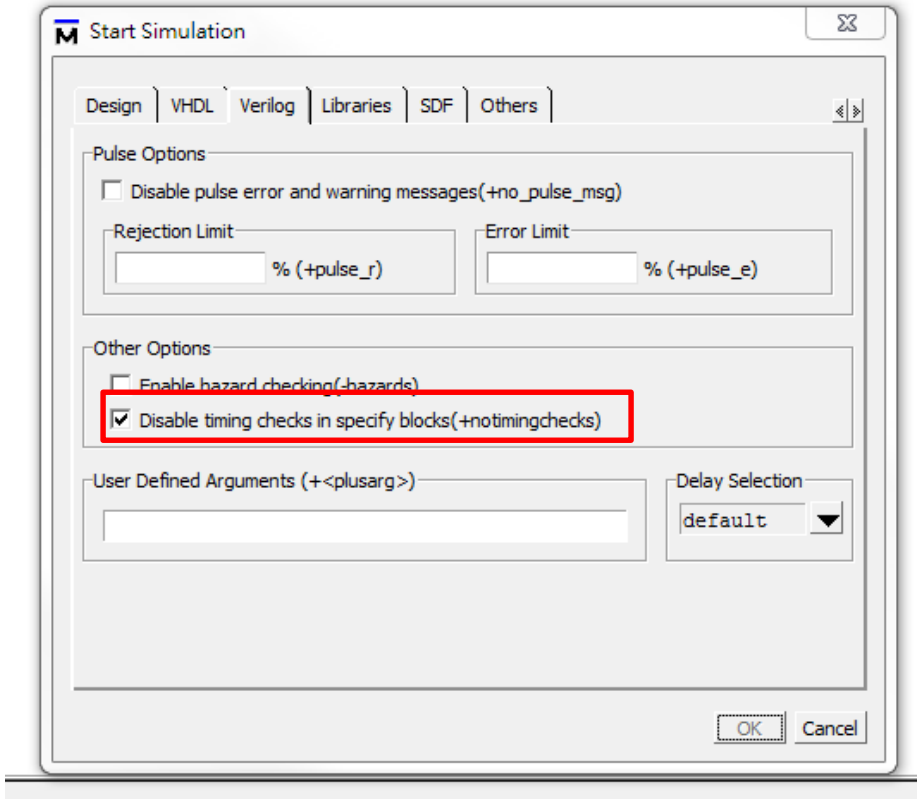






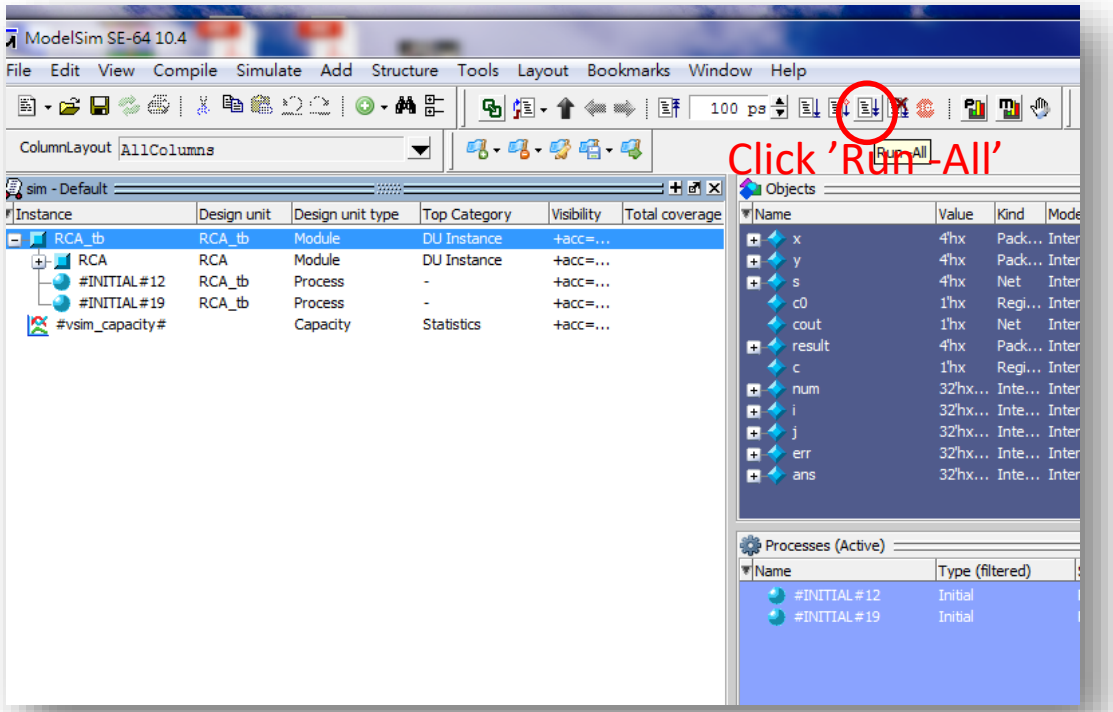
Functional Simulation by ModelSim

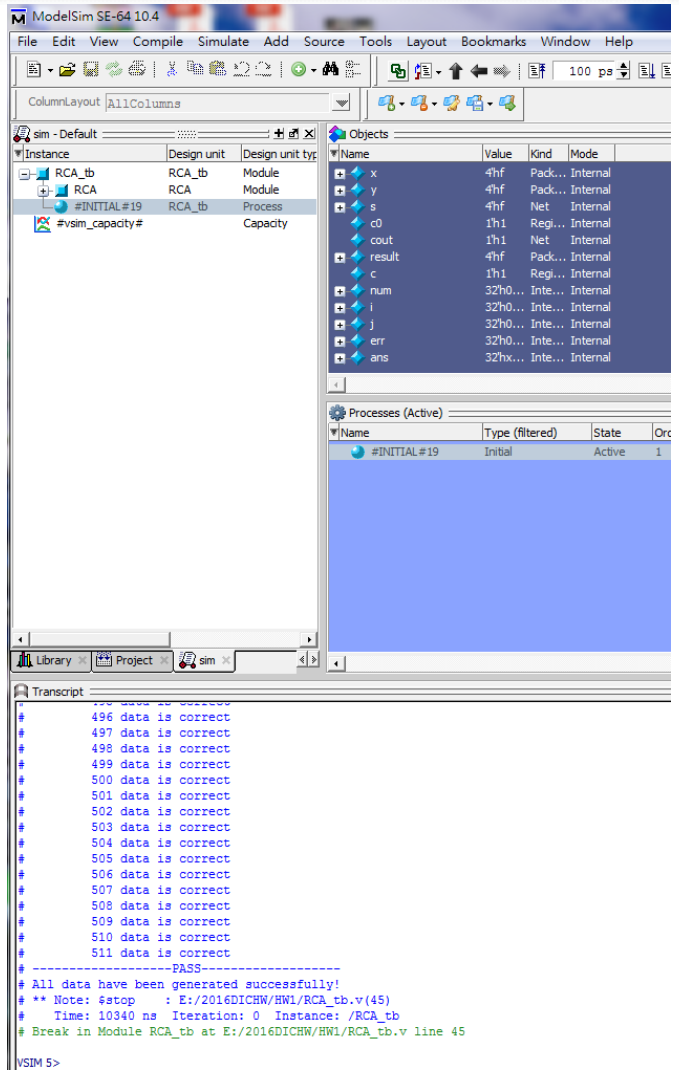
- Step 4: Start simulation
  - Check +notimingchecks

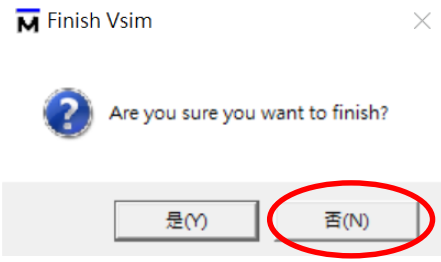


Functional Simulation by ModelSim

- Step 5 : Simulation





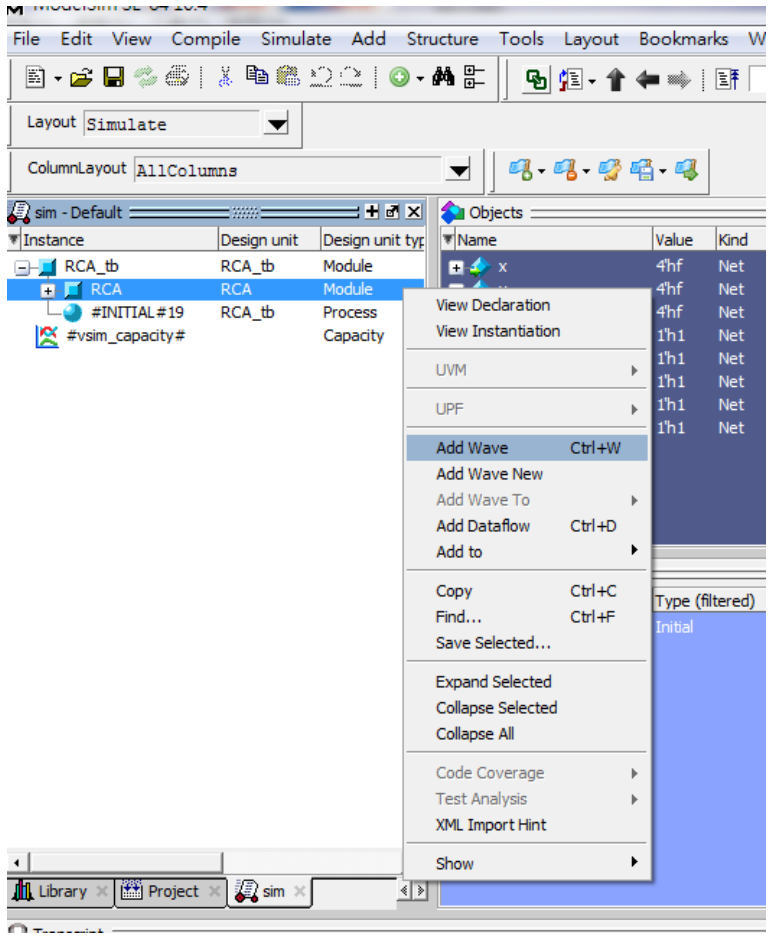


Click '否(N)'

*If you click '是(Y)', Modelsim will be closed

Functional Simulation by ModelSim

- Step 6 : Show waveform



You have to “restart” before a new simulation