ModelSim Simple Example

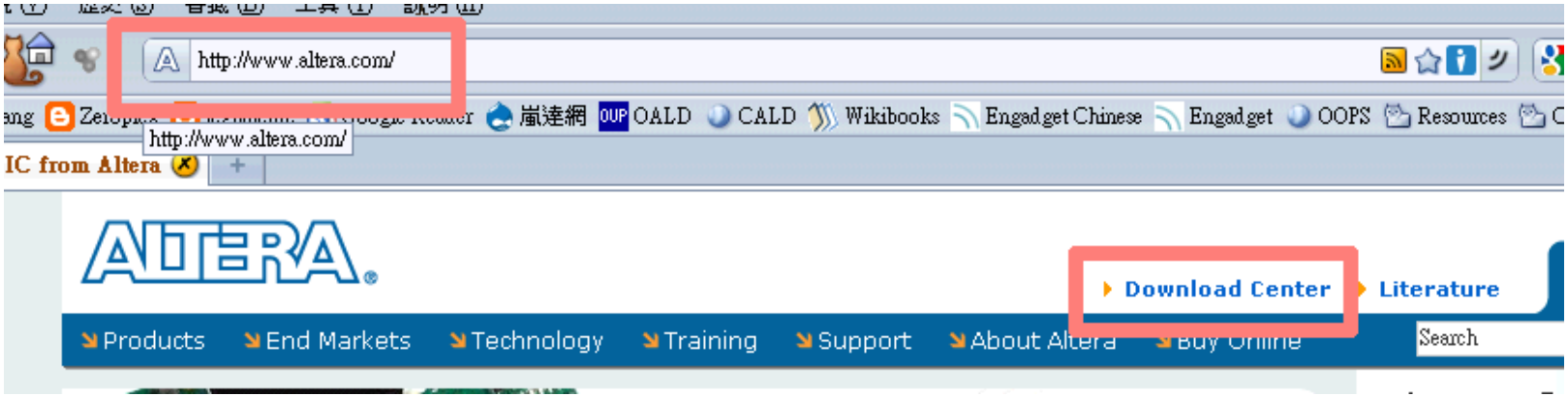
yen3

SoC Lab, CGUCSIE

January 5, 2010

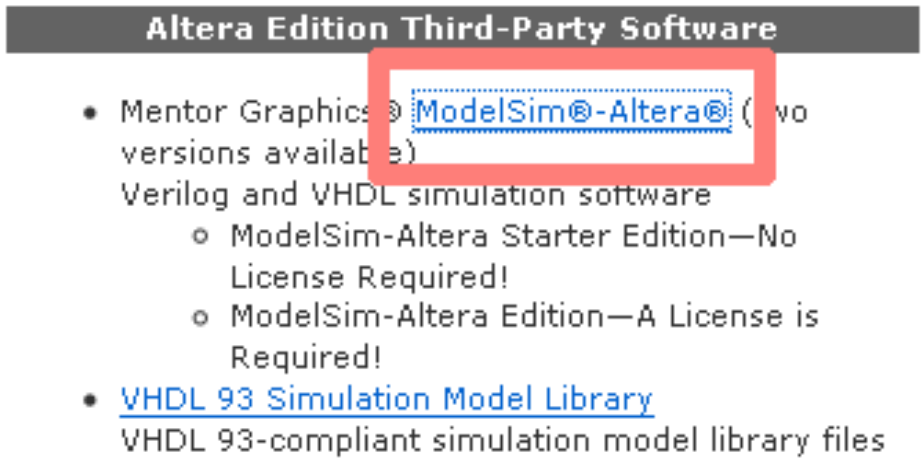
Download

- go to http://www.altera.com, and press “Download Center”



Download

- Press “ModelSim-Altera” in the “Altera Edition Third-Party Software” block



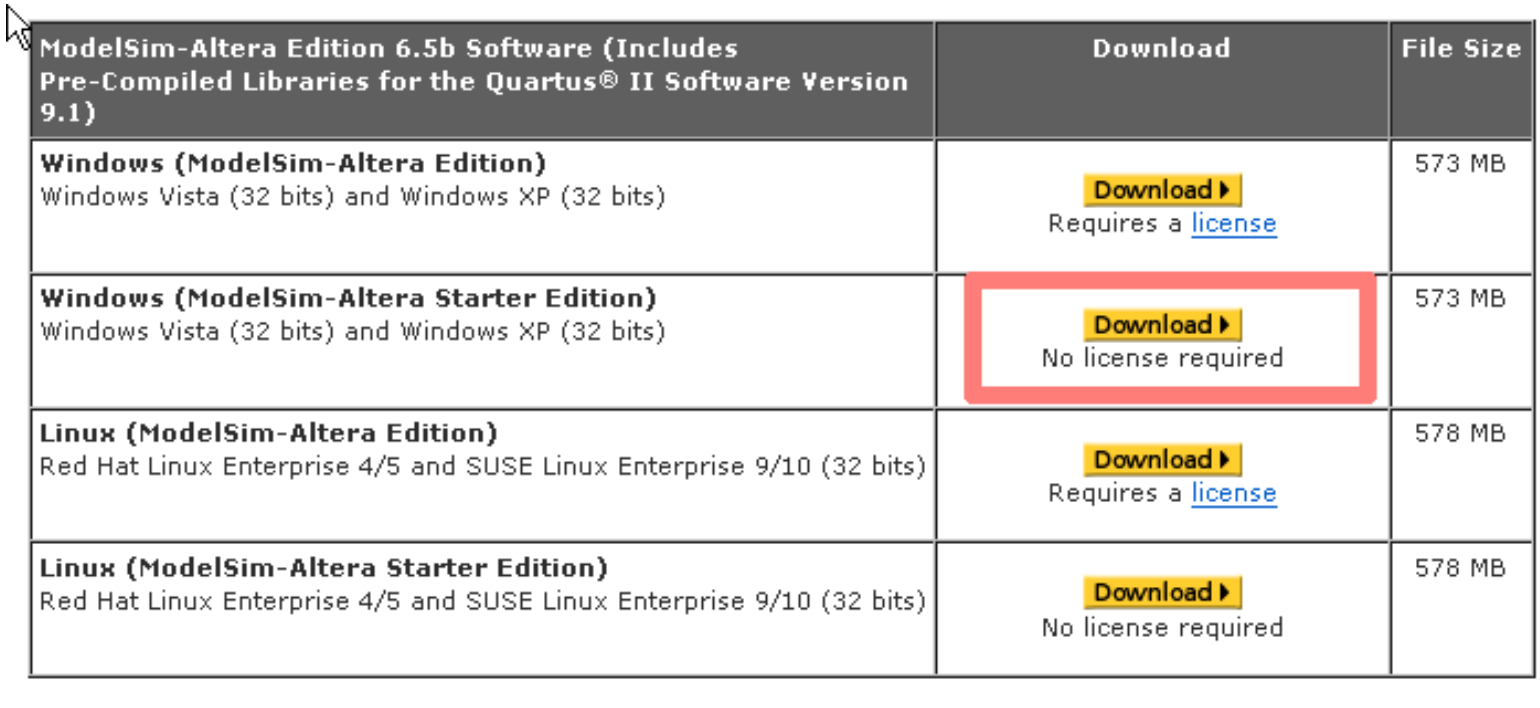
Download

- Press “6.4a” in the “ModelSim-Altera Starer Edition” block

Download ModelSim-Altera Software			
ModelSim-Altera Starter Edition NEW	**ModelSim-Altera Web Edition (1)**	**ModelSim-Altera Edition**	**Notes**
6.5b	—	6.5b	Use with Quartus II software version 9.1 only
6.4a	—	6.4a	Use with Quartus II software version 9.0 only

Download

- Press “Download” picture in the “ModelSim-Altera Starer Edition” block



ModelSim-Altera Edition 6.5b Software (Includes Pre-Compiled Libraries for the Quartus® II Software Version 9.1)	Download	File Size
Windows (ModelSim-Altera Edition) Windows Vista (32 bits) and Windows XP (32 bits)	Download Requires a license	573 MB
Windows (ModelSim-Altera Starter Edition) Windows Vista (32 bits) and Windows XP (32 bits)	Download No license required	573 MB
Linux (ModelSim-Altera Edition) Red Hat Linux Enterprise 4/5 and SUSE Linux Enterprise 9/10 (32 bits)	Download Requires a license	578 MB
Linux (ModelSim-Altera Starter Edition) Red Hat Linux Enterprise 4/5 and SUSE Linux Enterprise 9/10 (32 bits)	Download No license required	578 MB

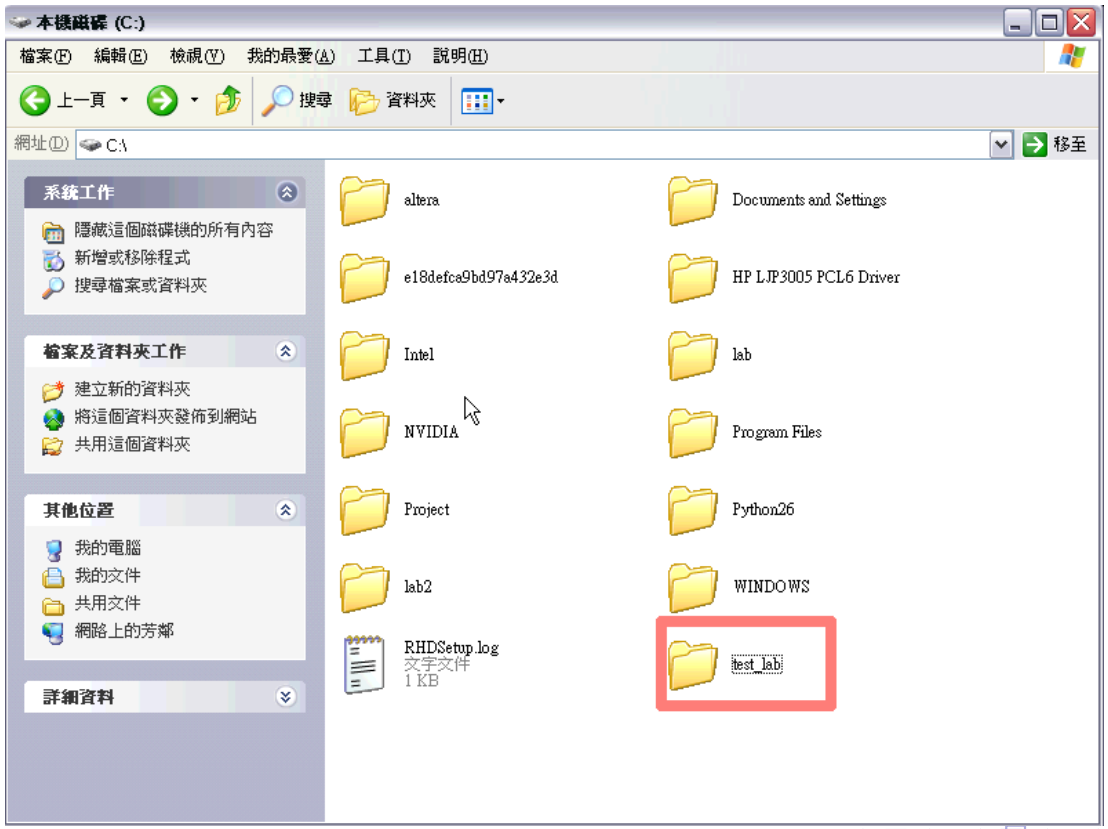
- The Next step is mentioned in “Quartus II 安裝與建置”

Install

請自行安裝 XD

建立資料夾

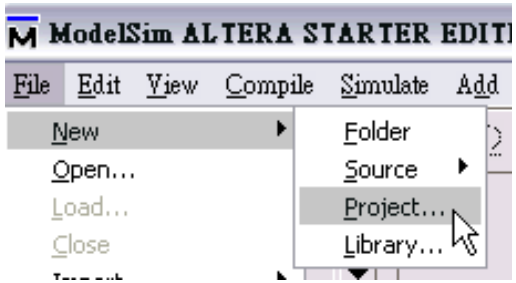
- 首先我們在 C:\ 底下建立一個 “test_lab”
- 注意: ModelSim 不接受中文檔名及路徑,
  所以千萬不要把資料放在桌面(桌面本身就含有中文路徑。)



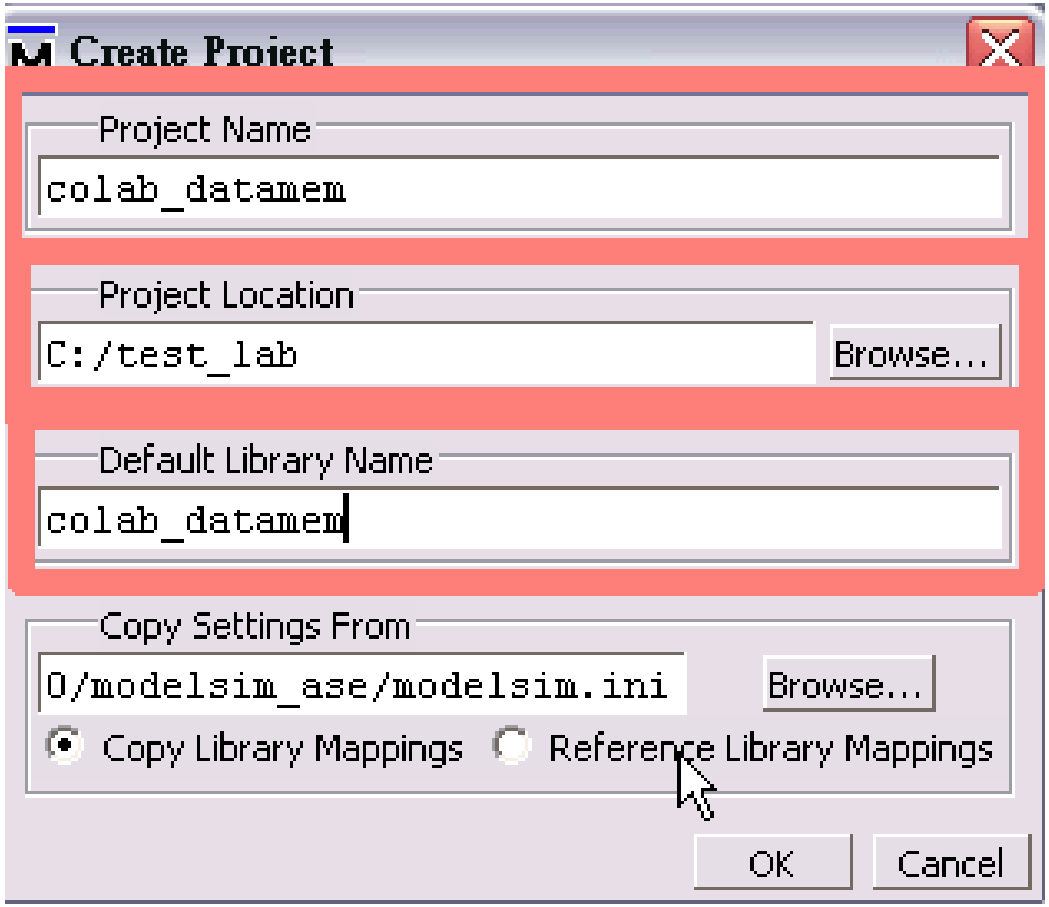
- copy file(`testbench.v`, `data_memory.v` `data.mem`) to the folder

- Open “ModelSim Starter Edition”

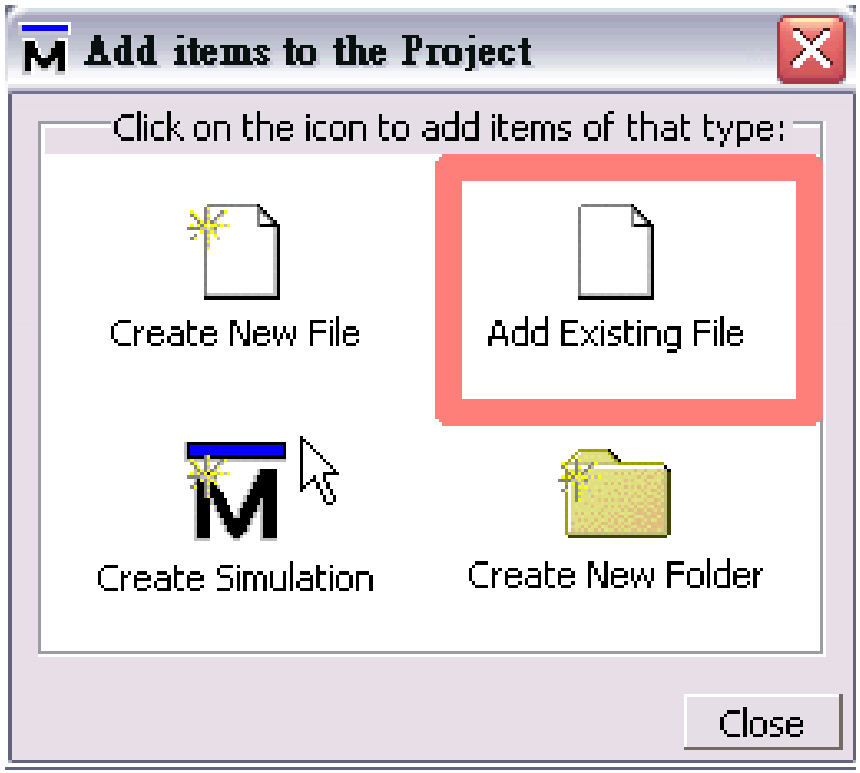
- Press “File” → “Project”



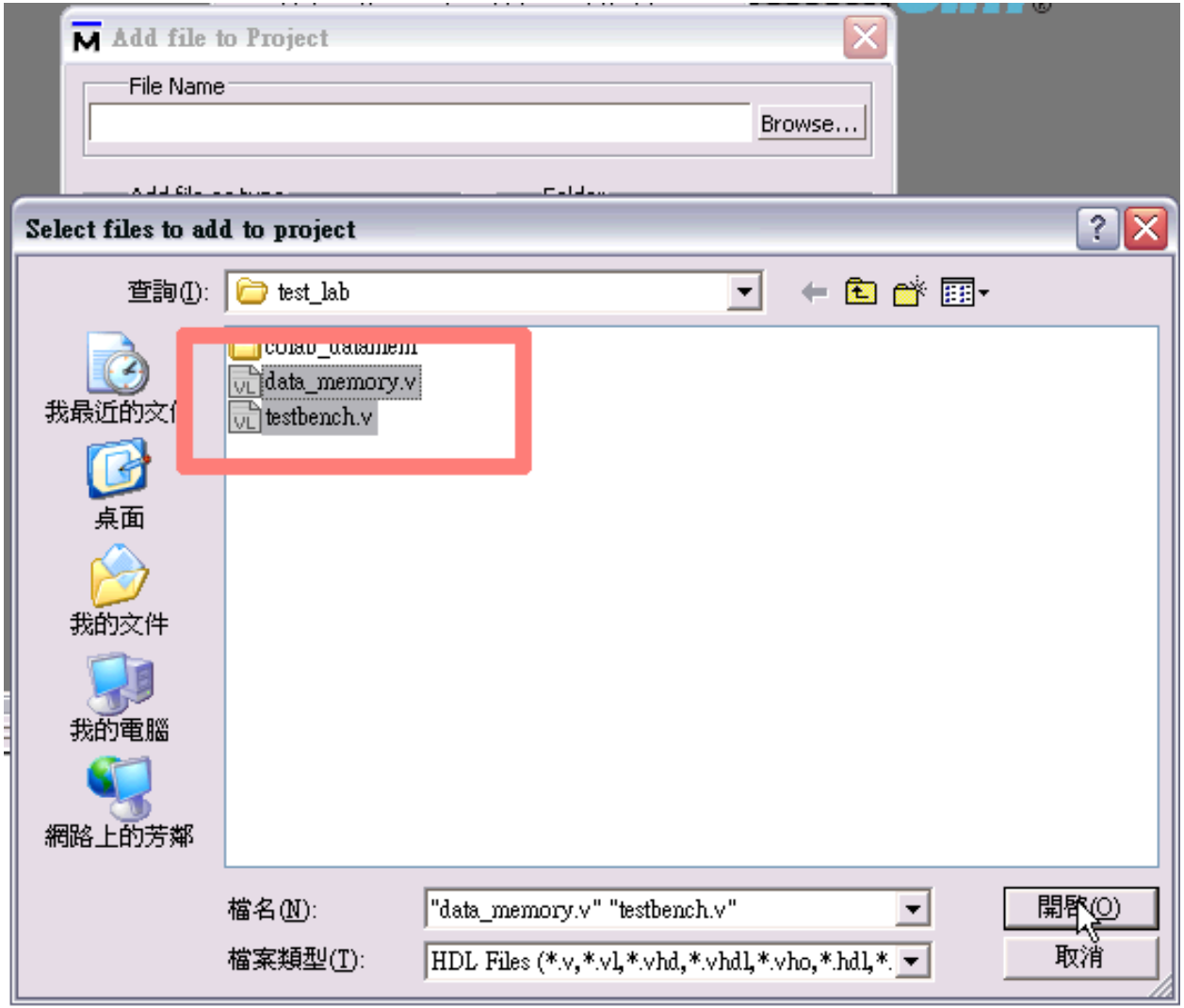
- Project Name — 不要亂取
- Project Location — 選取你剛剛所建立的資料夾
- Default Library — 取一個有意義的名字，等一下會用到



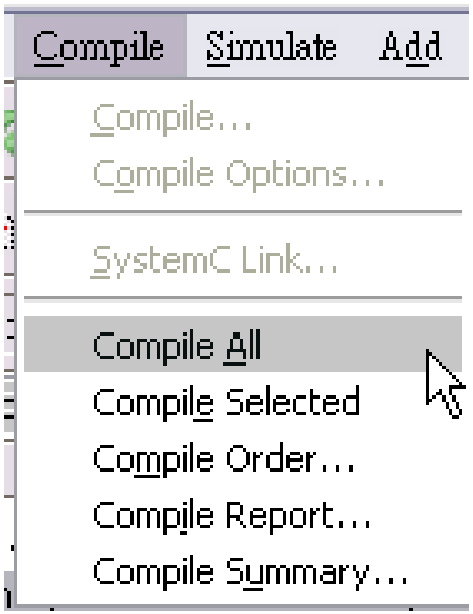
- Press “Add Existing File”



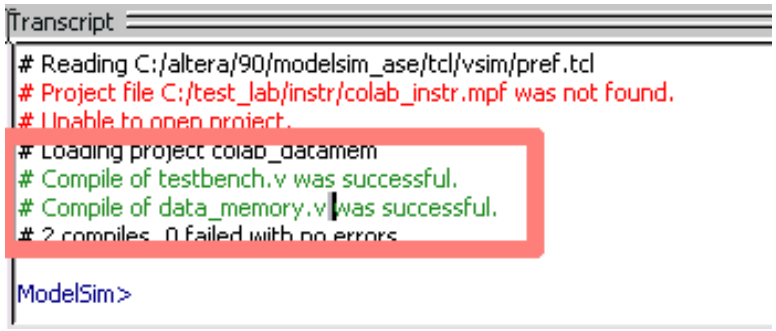
- Press “Browse”
- Choose the file what you copied `testbench.v, data_mem.v`



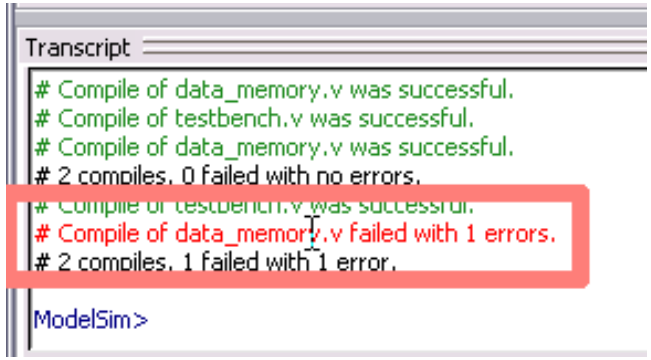
- Press “Compile” → “Compile All”



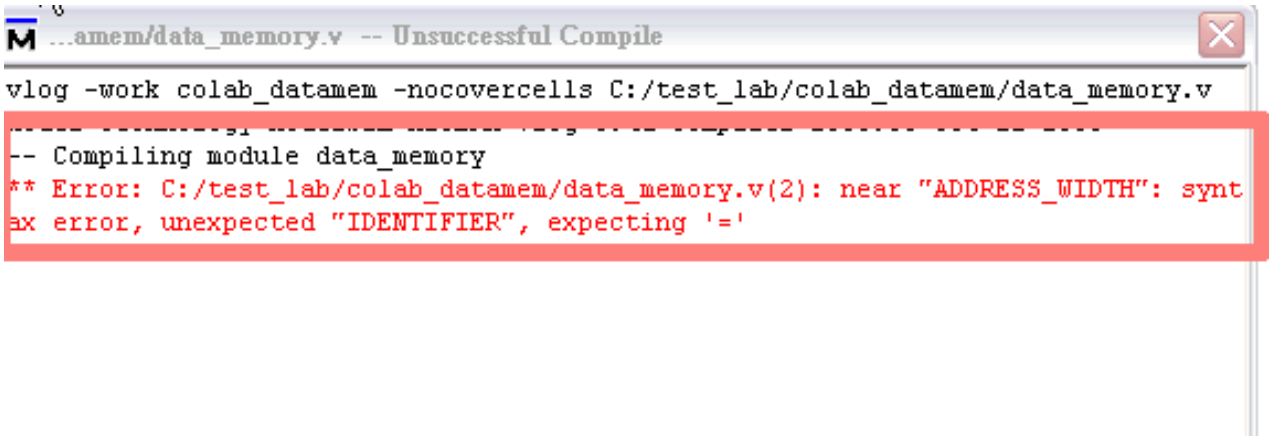
- You can see message window in the below. It will tell you compile result.
- You will see “sucessful”.



- If the program have compile error, it would appear Red word in the message window.

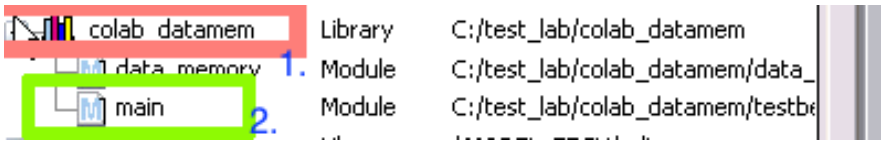


- You will see the error message after pressed the Red word

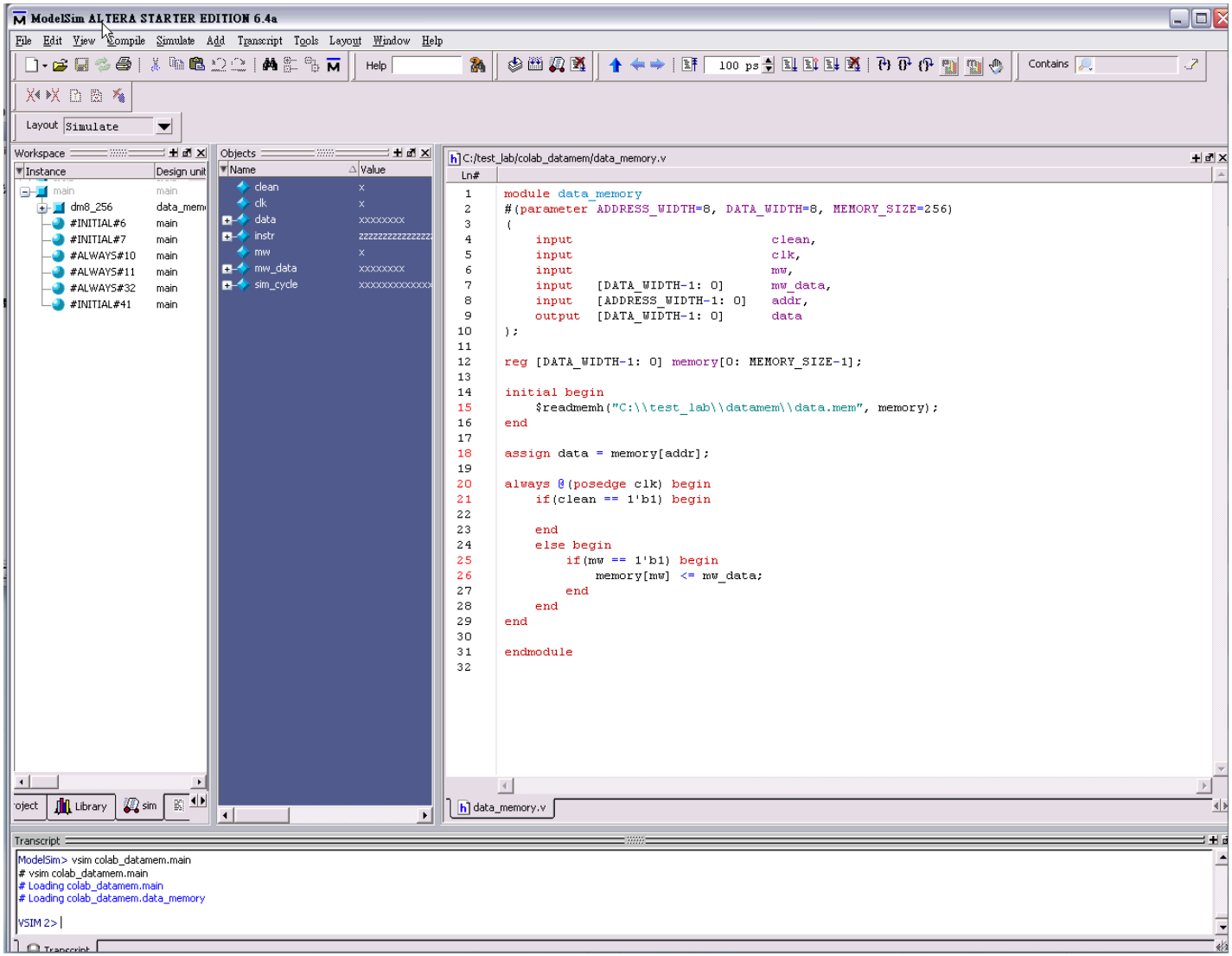


- Press “Library” tag page

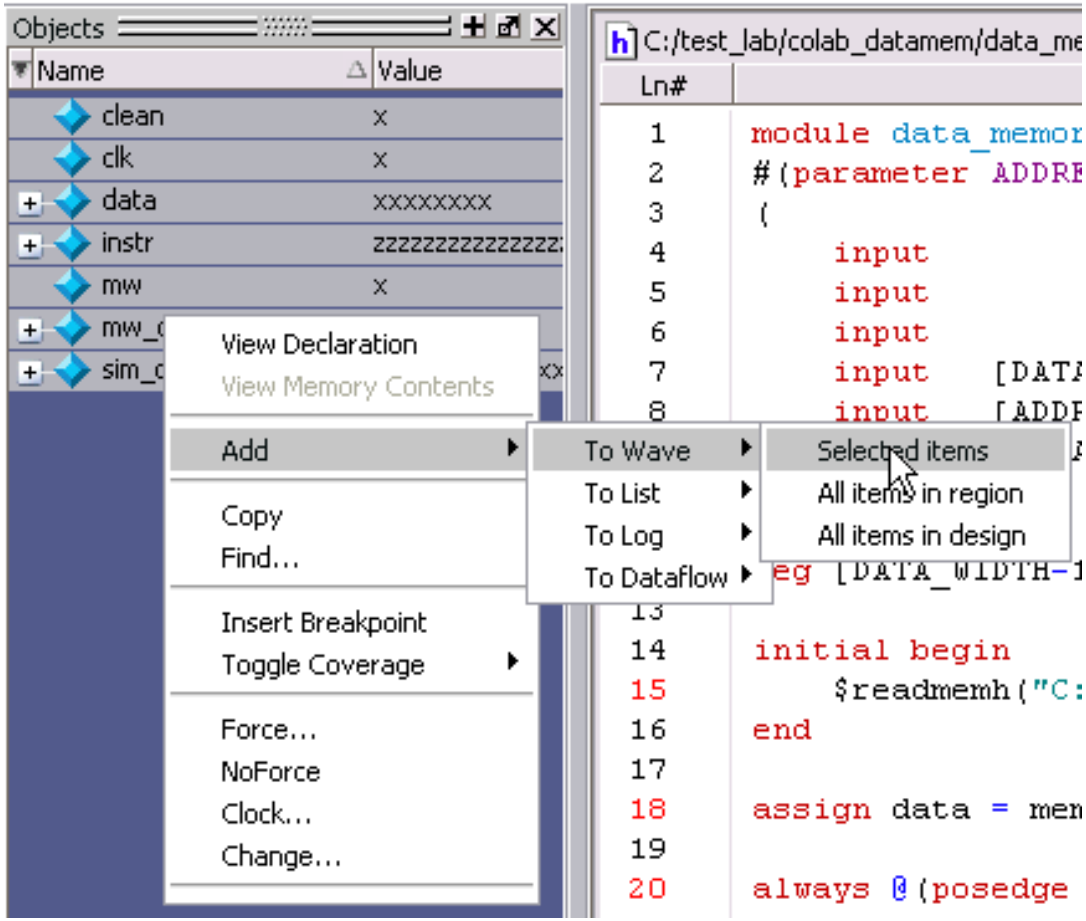
- find “colab_datamem”(your “Default Library Name”) and Press “main”



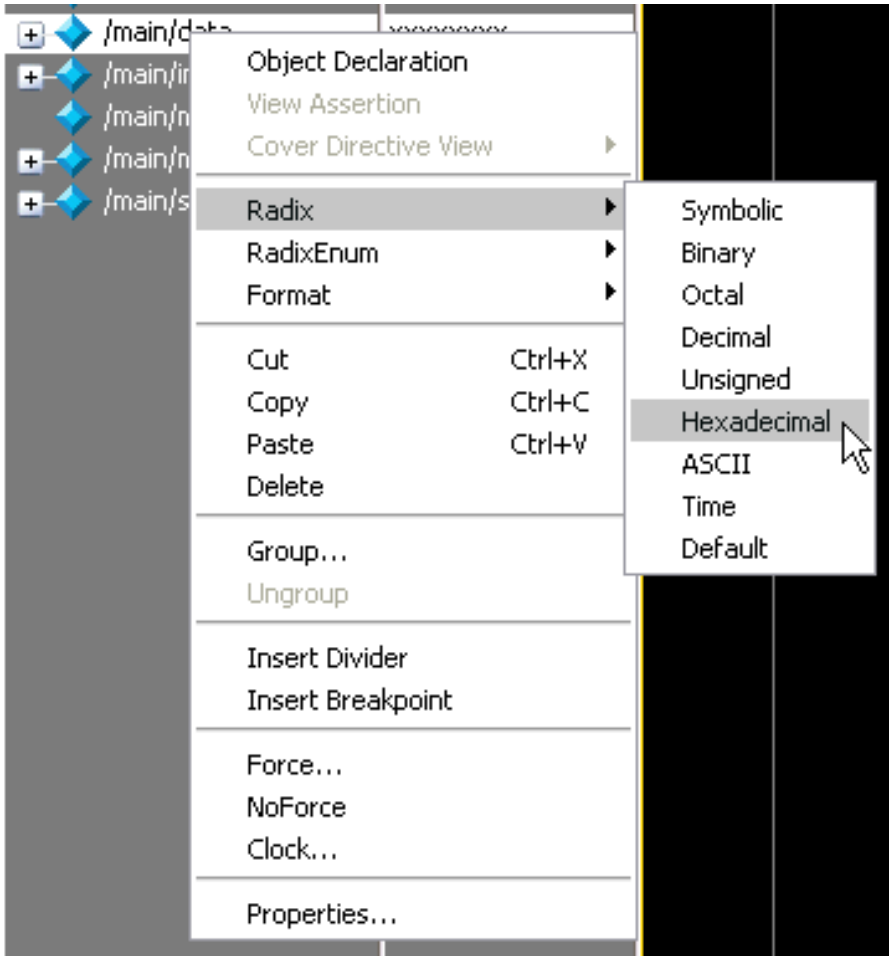
- 如果你看到這樣子的畫面，表示你快成功了。



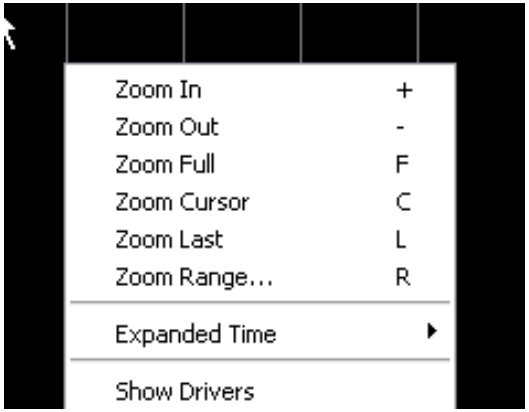
- 選擇你所要顯示的訊號加入 waveform



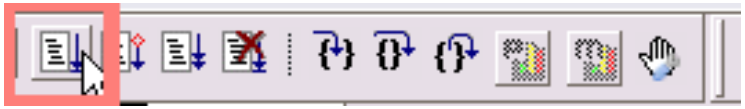
- every signal show in binary format, you can select the item and press right button of mouse. “Radix” → “Hexadecimal” or “Unsigned”



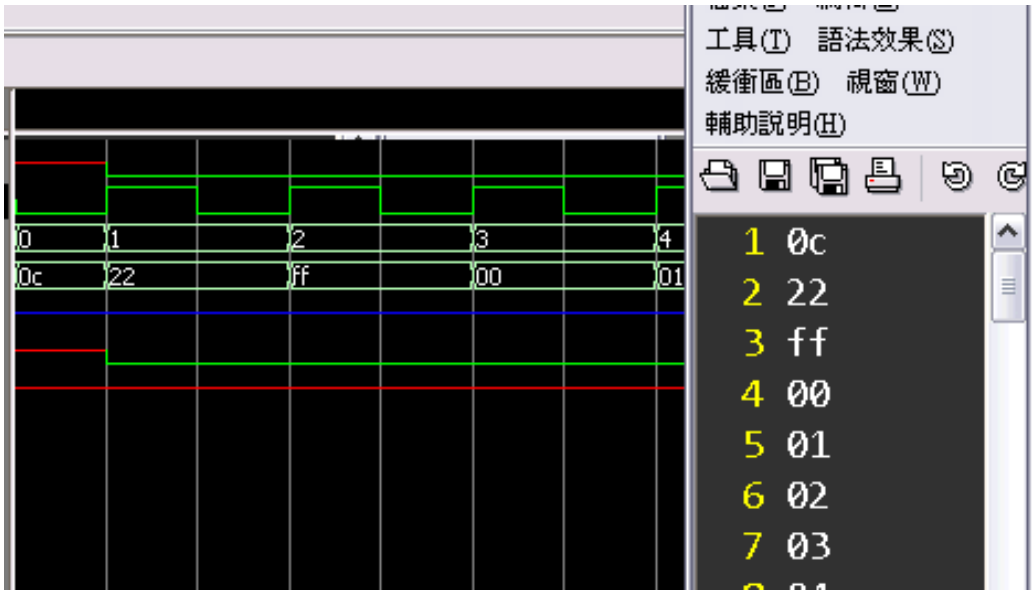
- Press right button of mouse in the waveform view, you can Zoom in it.
- A clock cycle's default length is 10ps.



- The program will start to simulate after press the button.



- You will see the simulate result in the waveform view



- If you set `$display()`, you will see the output in the message window.

Do you have any problem ?