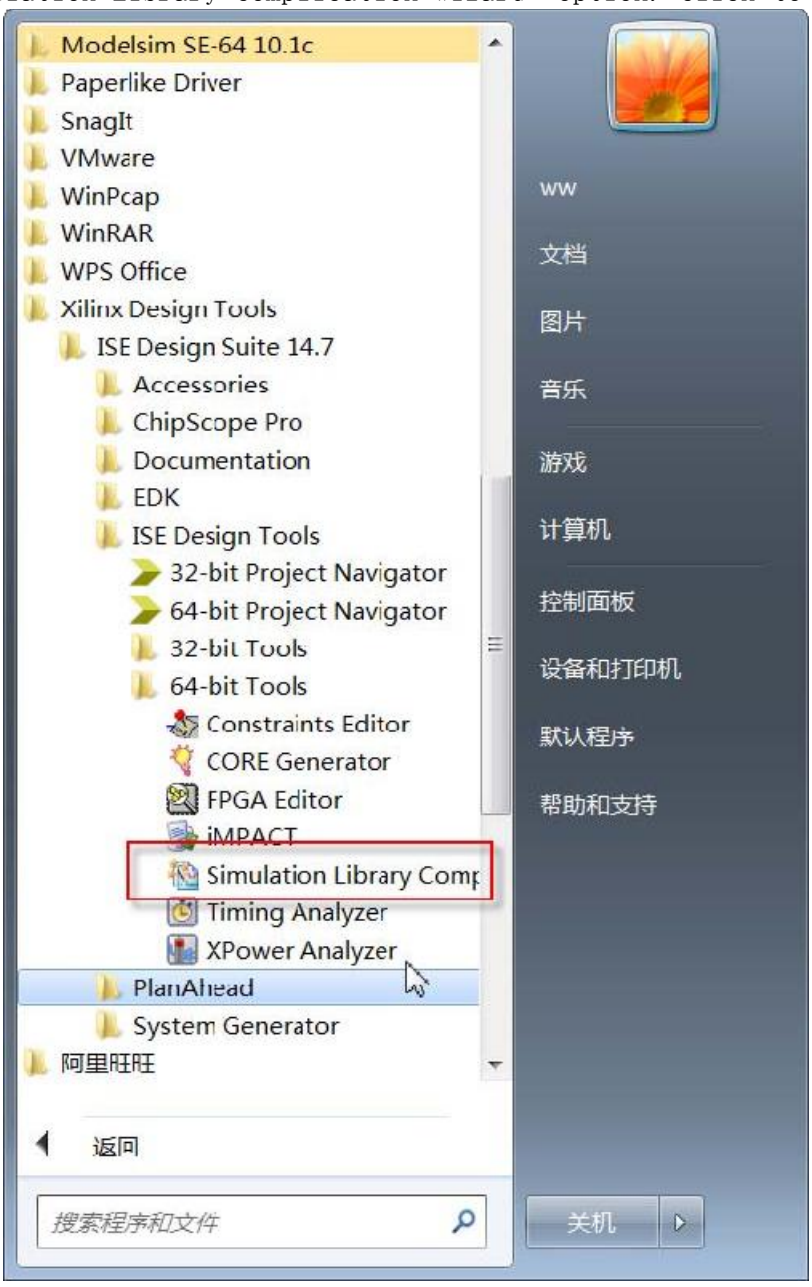
ISE association Modelsim settings

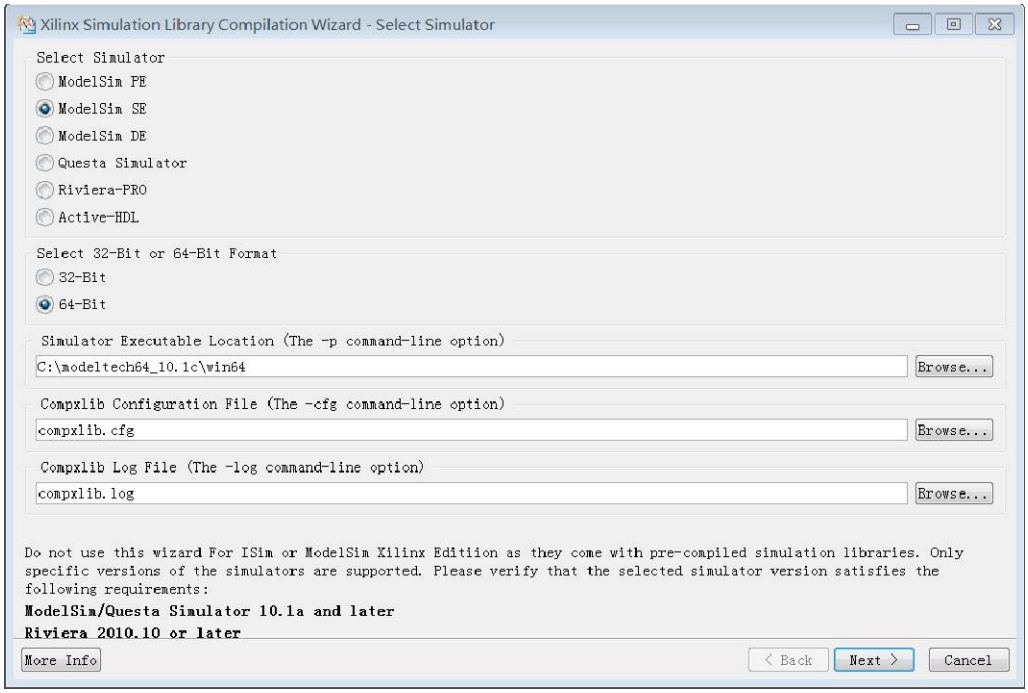
1. First generate the ISE simulation library file. In the Start menu, find the "Xilinx Design Tools-> ISE Design Suite 14.7->ISE Design Tools->64-bit Tools->Simulation Library Complication Wizard" option. Click to open



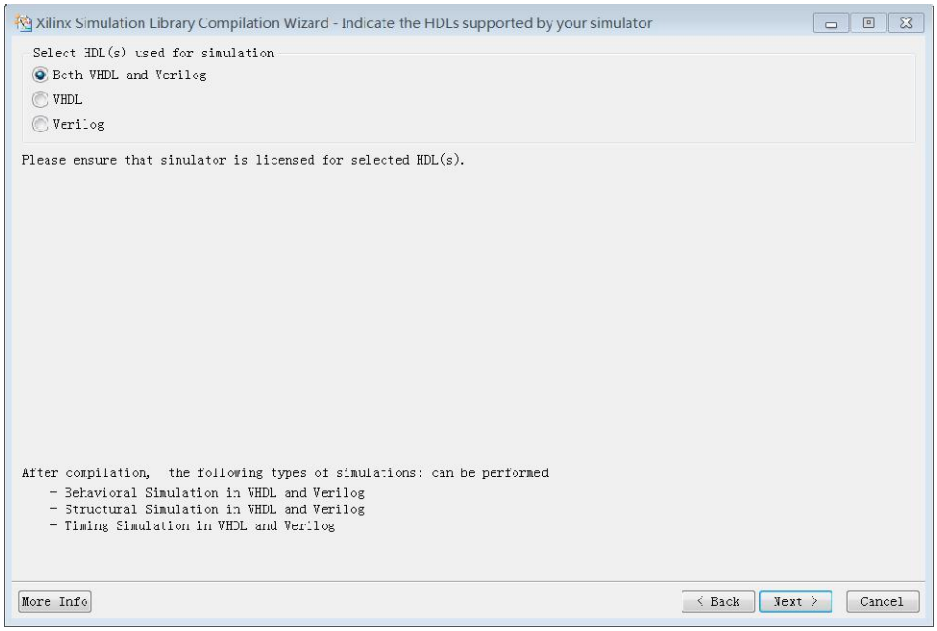
2. Select the Modelsim version as shown below, and we choose Modelsim SE here. Fill in the text of Modelsim.exe under "Simulator Executable Location"

3. Folder, click "Browse..." button, add, our Modelsim SE

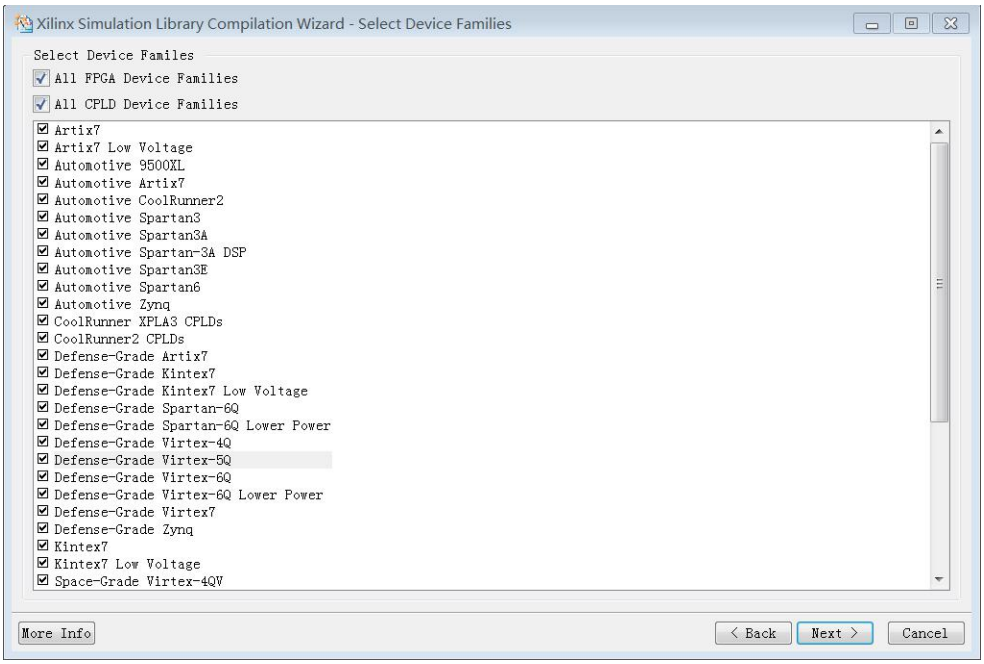
The path is "C:\modeltech64_10.1C\win64



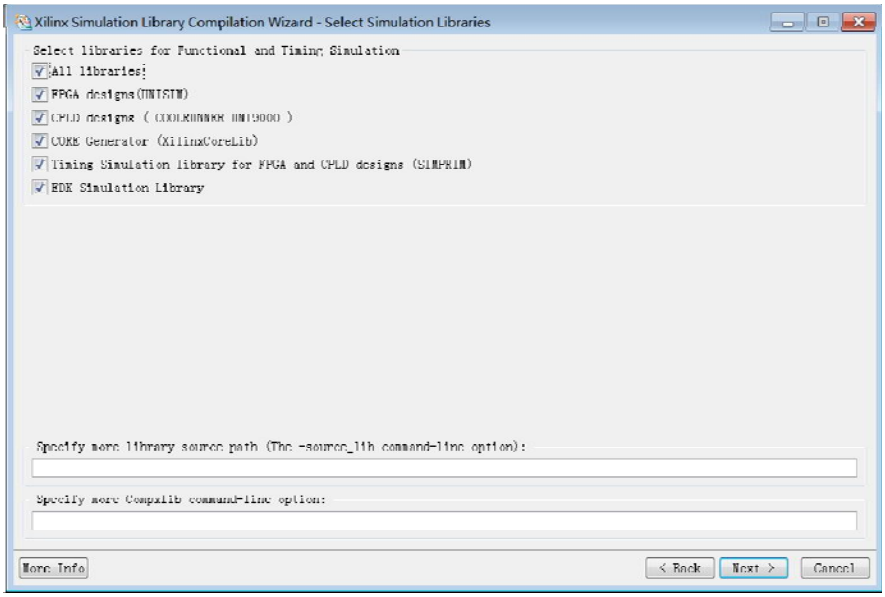
3. Select the language you want to compile as shown below. Generally we use the default option "Both VHDL and Verilog" and click Next.



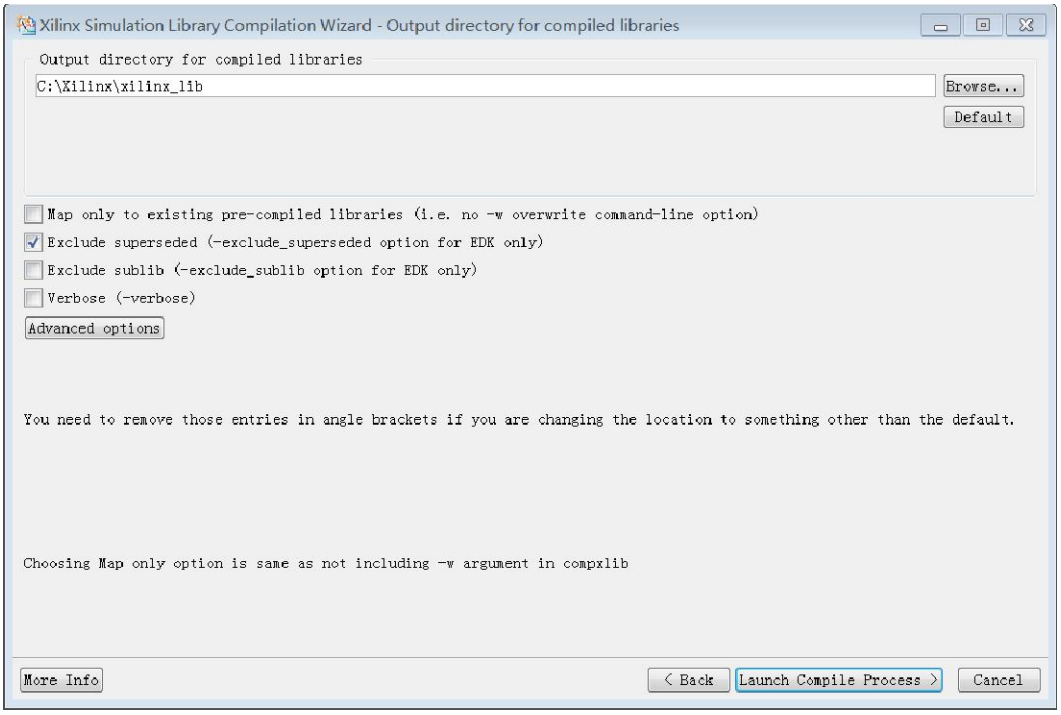
4. Select the Xilinx FPGA and CPLD device libraries that need to be compiled as shown below. The default is checked. Click "Next"



5. As shown in the figure below, this step is by default, all selected. The following two lines can be left blank, click "Next".

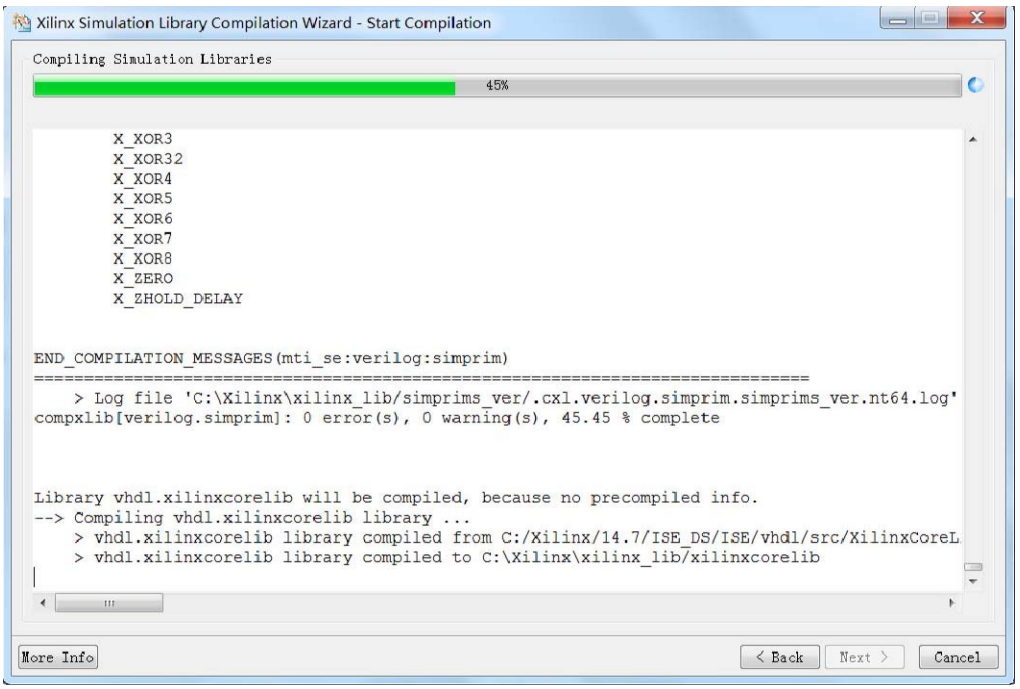


6. Fill in the path of the output compiled library under "Output directory for compiled libraries". Here we enter "C:\Xilinx\Xilinx_lib". Note that we need to create a new folder for Xilinx_lib in the C:\Xilinx directory. . Other options are also available by default, then click on “Launch Compiled Process”



7. Here, the key reminder: the compilation progress time is longer, depending on your computer configuration, my computer

It takes about 30 minutes, especially if the compilation progress will stay at 0% for a long time. This is normal. Please be patient.

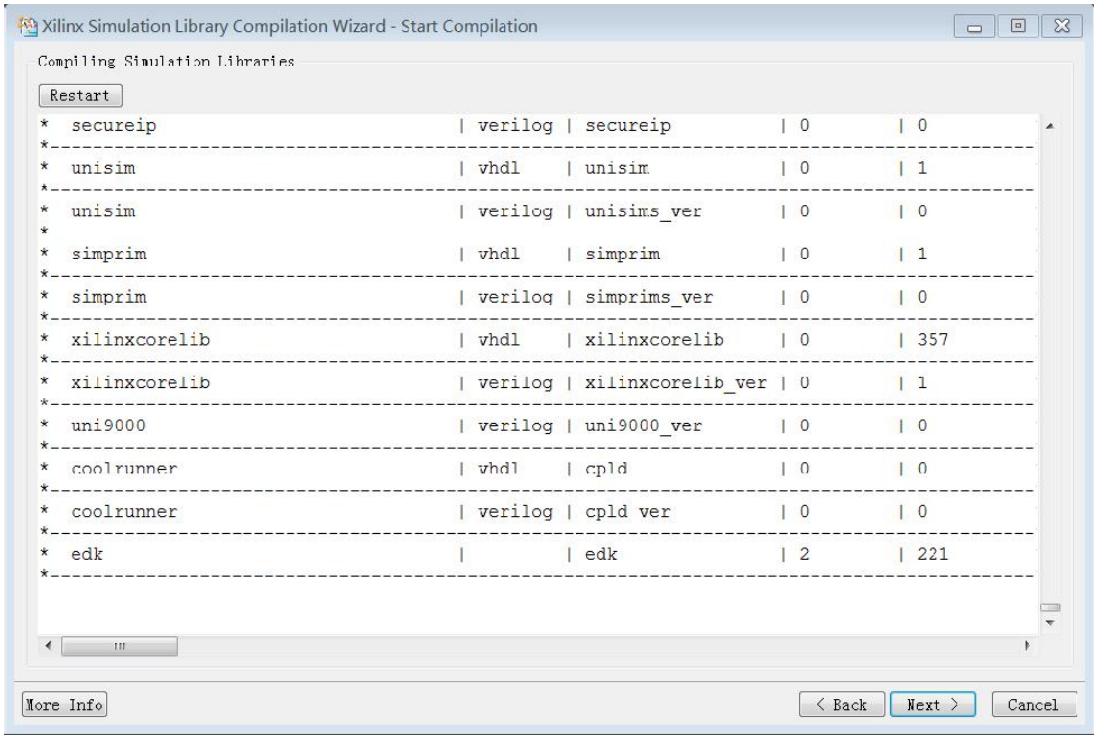


8. After 100%, it will jump to the interface as shown below, here report the error during compilation.

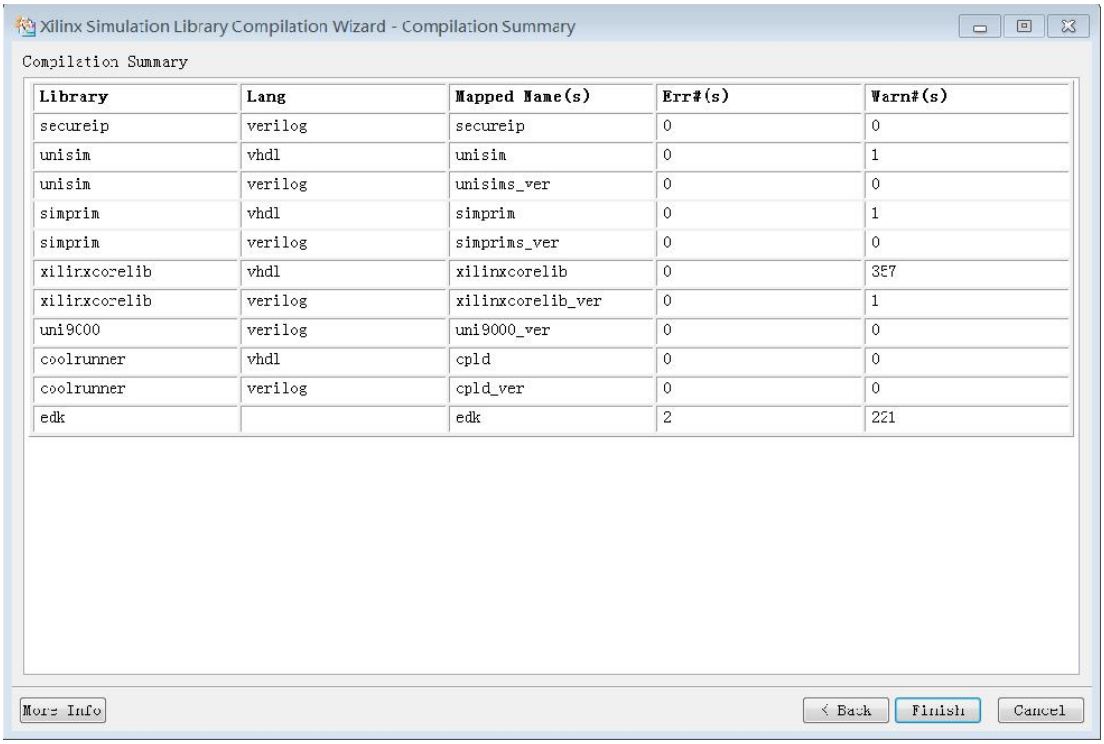
And warning, warning can be ignored, error must be looked at, if there is a

better return

See if the relevant path has Chinese or spaces, and the version is correct. I compiled edk here.
Error, can be ignored, we can not use it in the development process. Click "Next" to continue

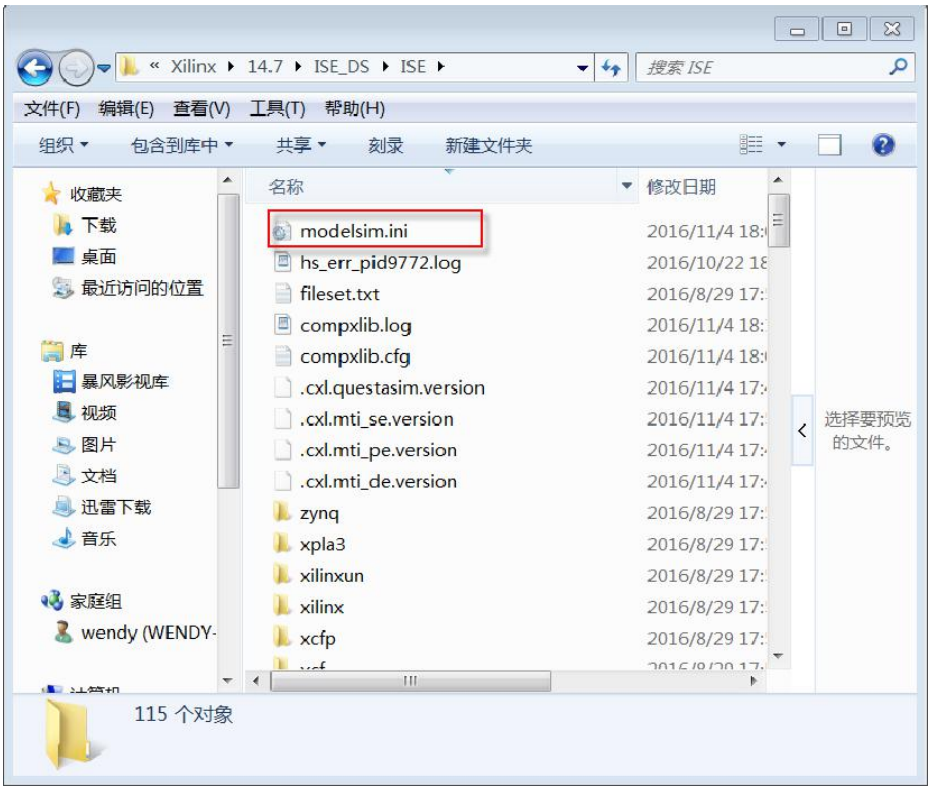


9. The final compilation report is shown in the figure below. Click "Finish" to complete the compilation of the entire device library.

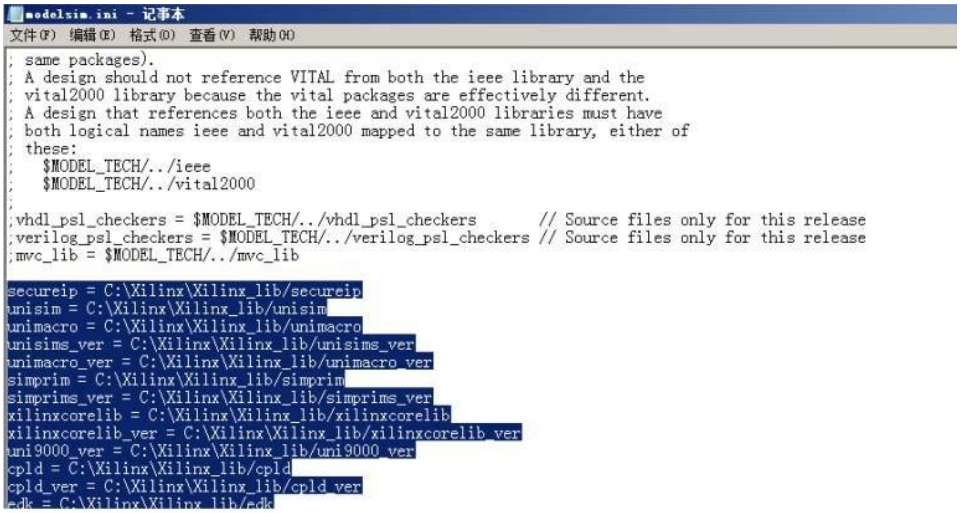


Library	Lang	Mapped Name(s)	Err#(s)	Warn#(s)
secureip	verilog	secureip	0	0
unisim	vhdl	unisim	0	1
unisim	verilog	unisims_ver	0	0
simprim	vhdl	simprim	0	1
simprim	verilog	simprims_ver	0	0
xilinxcorelib	vhdl	xilinxcorelib	0	357
xilinxcorelib	verilog	xilinxcorelib_ver	0	1
uni9000	verilog	uni9000_ver	0	0
coolrunner	vhdl	cpld	0	0
coolrunner	verilog	cpld_ver	0	0
edk		edk	2	221

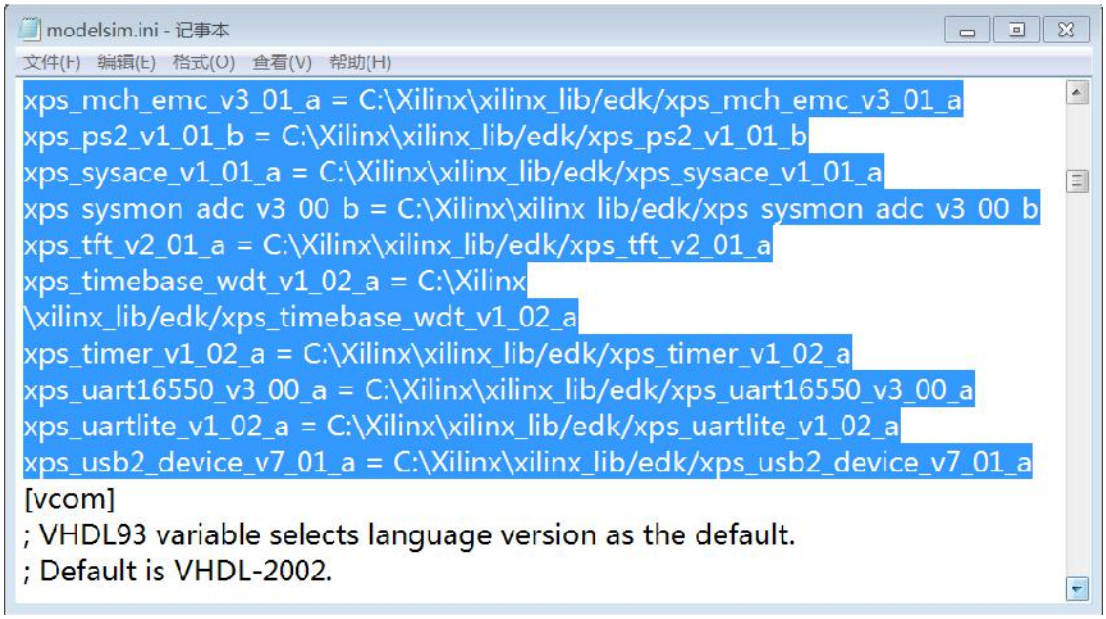
10. Now you will see a Modelsim.ini file in the ISE installation directory.



We open Modelsim.ini and copy the contents of the shadows (in the modelsim.ini file)
Line 47 starts from the top line of "[vcom]", that is, the content of the 308th action, all selected and copied)
The picture below starts at line 47:

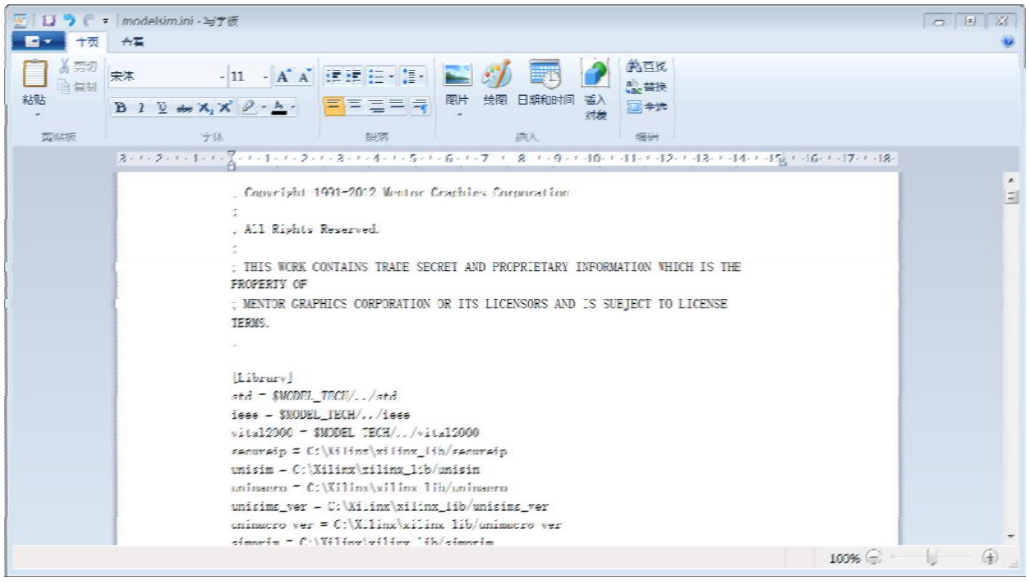


The following picture shows the behavior to 308

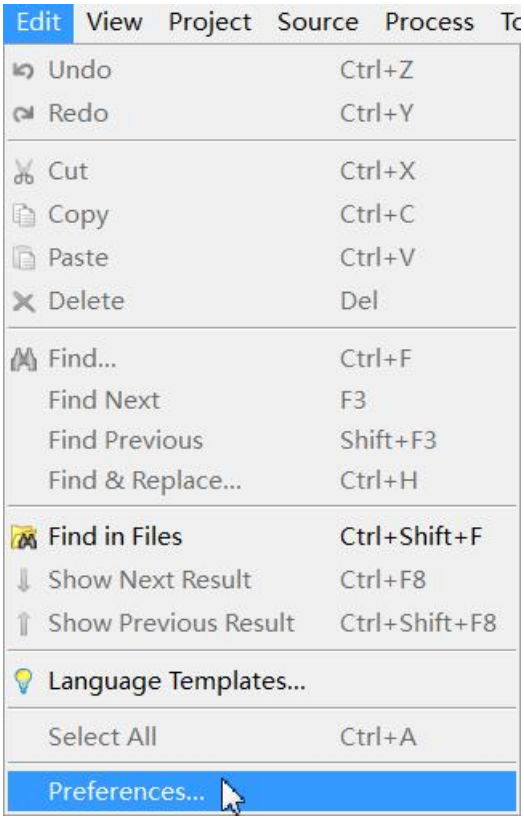


11. Under Modelsim's installation directory, ie "C:\modeltech_10.1c", look for

Open after "modelsim.ini" (note: first change the properties of this file under Modelsim - remove the read-only attribute). At the end of line 12, enter a line feed, and then paste the previously copied content as shown. Don't delete the original content.

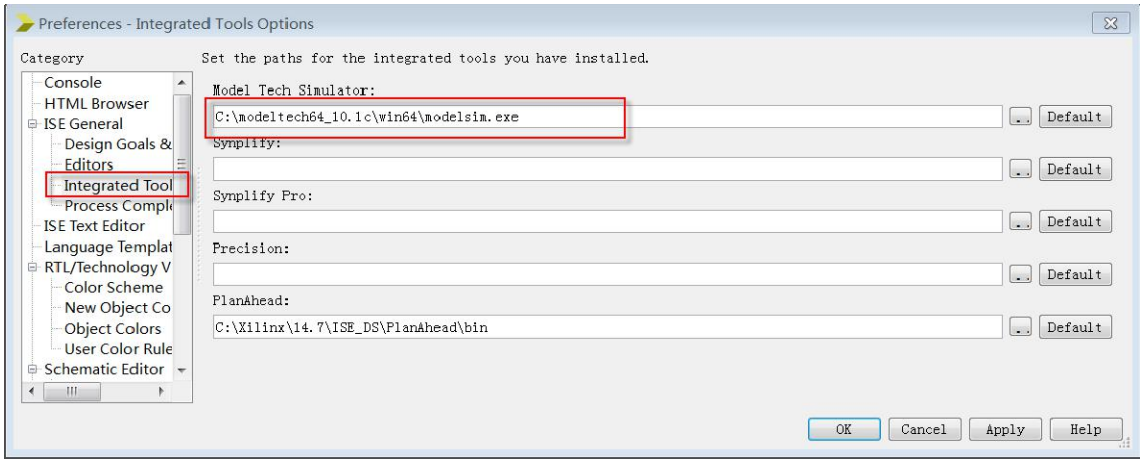


12. Open ISE14.7, then look for it on the ISE menu and click to open the Preferences window.



13. Under “Category” on the left, select “ISE General”->“Integrated”

Tools. In the Integrated Tools settings, enter the file path of Modelsim.exe under "Model Tech Simulator" on the right. Our path here is "C:\modeltech64_10.1c\win64\modelsim.exe" as shown. After completing the settings, click "OK"



At this point, the software installation is complete.