Altera Maxplus Tutorial

- This is a short tutorial on Altera Maxplus schematic capture and simulation
- You have the choice of using either the Unix workstations (Simrall 1st floor) or your PC.
  - I would suggest your own PC; no competition for seats, convenient, very fast if you have a 200 Mhz or better CPU and 32 Mb of memory.
  - Executing the file '/pc/SETUP.EXE' on the CDROM in your textbook will install Maxplus on your PC.
  - Files created under the PC version are compatible with the Unix version and vice-versa.
  - Appendix B in the back of your textbook has a very detailed tutorial and is the basis for Lab #0 in the course.

UNIX setup

- After you log in, to access the MAXPLUS software do:

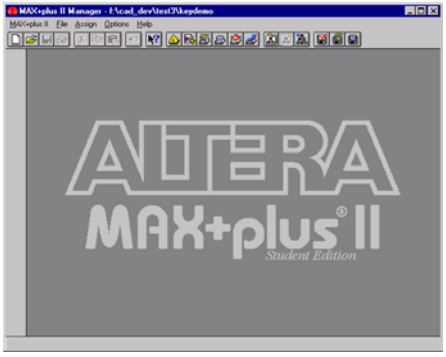
  % swsetup altera

  This will put the Altera binaries on your path list. This only needs to be done one time after logging in (or put in shell startup file).
- You should create a separate directory for each external assignment. The directory name is not important. The following unix commands create a directory, change into that directory, then startup maxplus

  ```
  % mkdir lab1
  % cd tut1
  % max2win
  ```

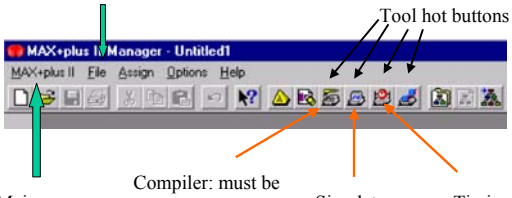
  The following slide shows the 'max2win' user interface.

MAX Plus Main Window



Main Menu Bar

Create new files, open old files, set project



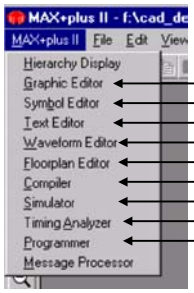
Main menu for accessing tools outside of quick buttons

Compiler: must be run on schematic or VHDL file before simulation

Simulator, waveform must have been created first.

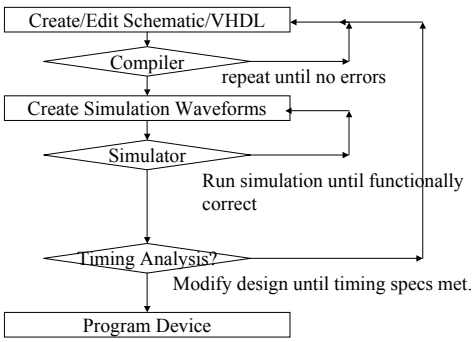
Timing analyzer

MAX+plusII Menu



Create new schematic
Rarely used, will use Auto symbol create
Edit VHDL files
Edit/Create waveforms for simulation
View logic mapping onto FPGA
Compile/Map schematic/VHDL to FPGA
Simulate Design
Analyze Timing of Mapped design
Dowload design to FPGA

Methodology



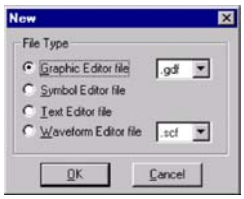
File Types

- .gdf Schematic Files (user created, schematic editor)
- .vhd VHDL Files (user created, text editor)
- .scf Waveform files (user created, waveform editor)
- .rpt Report of compilation process (tool created)
- .acf Project configuration file, automatically generated but can be edited by user (e.g., for pin assignments)
- .sym Symbol files, automatically generated, can edited by user (to create a custom symbol).

There are MANY, MANY other files automatically generated by various tools. Only the above types need to be preserved in order to keep your design; the other files can be deleted.

File Creation

To create any new file, use File -> New command from main menu, will pop up file creation menu, choose a type.



Compiler

After creating schematic or VHDL file, clicking on start button will start compilation process. After compilation is complete, can simulate design (if you have created a test waveform for the design).

For more detailed instructions

- Lab #1 has more detailed instructions in its writeup
- Experiment with different menu choices
- Look at online help
- Ask the TA

Sample Schematic

A sample schematic is show below for reference.