5 January 2004

Design Architect (DA) & QuickSim II Quick Reference Guide

Authors: David M. Sendek, Charles Thangaraj, Vinil Varghese, Medha Kulkarni and Tom Aurand

Background: Mentor Graphics DA is a schematic capturing Computer Aided Design (CAD) tool for use with digital logic gates and/or analog components. QuickSim is Mentor Graphics' digital simulator. The circuit is simulated by forcing input values and then observing the results graphically. *This handout is intended as an introduction to DA & QuickSim II. Through experimentation, trail-and-error or reading the Mentor Graphics Manuals, your ability will expand with this CAD tool.*

References:
(a) "Using Mentor Graphics for Digital Simulation", David M. Zar, 9/1/97, http://ge.ee.wustl.edu/dzar/tutorials/digital
(b) "Force File and Force Commands", David M. Sendek, 9 July 2003

Assumptions:

1. Student has an account on the engr server
   Note: If not, student must go to Anderson Lab in Glover to create an account.
2. Student knows how to sign-on the engr server.
3. Student knows how to go from a Windows to Unix environment.
4. Student has basic understanding of Windows OS and Unix commands.

SETTING UNIX ENVIRONMENT FOR DA & QuickSim (steps at the Unix prompt)

1. Create a directory ee451, if one does not already exist
2. Create a sub-directory for each lab, ex: lab2, lab3, …
3. Create sub-directories under each lab, as appropriate.
4. **%setenv MGC_WD $HOME/path**
   ex: %setenv MGC_WD $HOME/ee451
5. Change to your working ee451 sub-directory
6. **%source /mentor/csu_setup/idea_setup.csu**

Notes:

- There is a **space** between %source and /mentor…
- This permits you to gain access to both DA & Quicksim
- If you do not modify your .cshrc file, you must repeat steps 3 through 5 each time you re-open your unix window.

FEATURES OF DA

STARTING & EXITING DA

1. Starting DA:
   - At Unix prompt, enter %da
   - Once da has launched, maximize the screen. Clicking on the "large block" in your upper right hand screen.
2. Exiting DA: select MGC > Exit

**** Note: To drill-down a menu select the "Right Arrow" must be selected (left mouse click)

CREATING, OPENING, SAVING AND CHECKING A (DESIGN) SHEET

1. Creating a Sheet (steps):
   - In the "session_palette" window click on the icon "OPEN SHEET".
   - Click OK.

   Note: Your default sheet will be sheet1
2. Opening a Sheet (steps):
   - Under the "session_palette" window, click on the icon "OPEN SHEET".
   - In the "Open Sheet" pop-up screen, click Navigator …
   - In the "Component, Schematic, or Sheet" pop-up screen, click on the Component Name you provided when you first saved the circuit diagram.
   - In "Open Sheet" window, select OK
3. Checking your Design (steps):
   - In the "Design Architect" window, Check > Sheet > With Defaults

     Notes:
     - Checking your sheet ensures that you produce a valid, workable circuit.
     - You will not be able to save and use the logic as a symbol, if you do not check the sheet.
     - To find the location of errors or warnings: Select Edit > Select > By handle. The "Select By Handle" pop-up window will be displayed. Enter the handle/signal name. ex: N$9
4. Saving a sheet (steps):
   - Select File > Save Sheet as
   - At the "Save Sheet As" screen, in the "Component Name" entry, add the name of your circuit diagram.
   - In pop-up "Save Sheet As" screen, enter sheet name: Ex: sheet1, sheet2 … (sheet1 is your default name)
   - Click OK

   *** Note: Each sheet should be in its own subdirectory
   EX: …/ee451/lab2/1bit
   …/ee451/lab2/3bit

***** **Make sure you save your design in the directory you want it in** *****

SELECTING A COMPONENT LIBRARY (steps)

1. In the "session_palette" window, click on the icon "LIBRARY"
2. Select, gen_lib (this is the library we will be using in EE451). Another useful library is the Generic_Lib. This can be used by selecting Libraries > MGC Analog Libraries > Display Libraries Palette. In the MGC Analog Libraries window, select Generic_lib
3. If a scroll bar is not present with the gen_lib, place the mouse arrow over the gen_lib. On the mouse, hold and right click. Select "Show Scroll Bar"
4. Now you are ready to start adding components to your sheet.

ADD & DELETING LOGIC SYMBOLS AND COMPONENTS

1. Adding Components:
   - In "gen_lib" window, select the various gates required for your circuit design.
   - The symbol selected will be displayed in the window above the "gen_lib" window.
   - A portin, portout or portbi components for I/O terminations are found in gen_lib.
   - There may be a need for transistors or ground signals. These can be found in the analog library.

   Notes:
   - Use your mouse to drag the symbol to your desired location.
   - Use the left mouse button to set into position the logic symbol
2. Deleting Components:
   - With the left mouse button, select the symbol
   - *The symbol(s) to be deleted will now be drawn in dashed-lines*
   - Click the right mouse button. The "Instance" pop-up window will be displayed. Select "Delete"

   Notes: After every add or delete, the symbol selection list needs to be cleared. Use F2 to unselect the object so that another symbol can be added/deleted.

ADDING & DELETING WIRES

1. To add a wire (steps & notes):
   - Ensure the "session_palette" window is displayed. In the "Design Architect" window, select Libraries > Display Schematic Palette
   - In the "session_palette" window, click on the icon "ADD WIRE".
   - Shortcut: F3

   Notes:
   - Left mouse click at the starting point
   - Left click on the mouse, to create vertices, if required.
   - End wire connection by double-clicking on the left mouse. This can also create a connection point.
   - To get out of the "ADD WIRE" mode, click Cancel in the "ADD WIRE" pop-up window.

2. To delete a wire (steps):
   - Use F2 to unselect wires/nets & logic
   - With the left mouse button, select the wire
   - *The wire(s) to be deleted will now be drawn in dashed-lines*
   - Click the right mouse button. The "Net" pop-up window will be displayed. Select "Delete"
3. To automatically connect wires (steps)
   - Select the wire
   - Shortcut: ctrl + F6

TO UNDO THE LAST COMMAND (In the "Design Architect" window)

- select, Edit >Undo

ZOOM IN/OUT (In the "Design Architect" window)

1. Zoom In: select View > Zoom In > *selectparameter*
   Shortcut: +
2. Zoom Out: select View > Zoom Out > *selectparameter*
   Shortcut: -

ADDING & DELETING LABELS

1. Add a portin or portout symbol to your wire.
2. To add a label:

- Select the portin or portout symbol
- Place the mouse pointer over the text "NET"
- Select shift+F7
- Enter the name desired in the new value box in lieu of the name "NET"

To delete a label :

- Select the port. (left mouse click)
- Right mouse click. Select delete

ADDING & MODIFYING INPUT/OUTPUT PORTS (NETS)

1. Select the port you want to Name (left mouse click)
2. In the "Design Architect" window, select NET > Name

PRINTING SCHEMATICS

Select file > Print Sheet: Enter mentor_c207 as the printer name.

MAKING A "SYMBOL"

Background: A symbol permits you to group logic gates into a single symbol. The symbol then can be used in a subsequent sheet. This creates a hierarchical structure to your drawings. As an example: With a 3-bit Arithmetic Logic Unit (ALU) in lab 2, you can develop a 1-bit slice ALU. Then a symbol is created using the 1-bit slice ALU logic. In a subsequent sheet, three 1-bit slices (the symbol created) are then wired together.

*** **Caution:** To avoid problems, no more than one sheet and 1 symbol should be in a directory. You must control where the sheet and symbol is saved. Don't assume Mentor Graphics is handling this properly. This will require a directory tree structure for each lab. It is recommended that you use "save as" instead of "save" to ensure the file (the sheet or symbol) is saved in the proper directory.

1. Check the sheet prior to creating a symbol
2. In DA, select MISC > Generate Symbol
   - Select Sort Pins Yes
   - Click OK

   Notes: Do not make any additional changes to the "Generate Symbol" pop-up window.
3. In DA, select check > With Defaults
4. In DA, select File > Save Symbol > Default Registration
5. Open a new sheet
6. "Choose Symbol" icon to pull-in the symbol created

MAKING A NET ("Bus" or "Bundle")

1. Ensure you label your portin, portout or portbi like "A[2:0]", representing 3-bits
2. To "add" a net in DA,
   select miscellaneous > General Symbol
   select Yes for "Sort pins"
3. To "break-out" a signal from a bus:
   - Draw a wire
   - Once completed with the wire, the pop-up window "Choose Bus Bit" will be displayed.
   - Enter the bit you want to tap-off.
4. To add a bus,
   In "schematic_add_route" window, select icon "ADD BUS/BUNDLE"

SIMULATING YOUR DIGITAL DESIGN

STARTING & EXITING QUICKSIM

1. You need to be in a directory level above where your design directory is located. Note: If you use the default name "sheet1", your design name is the name of the directory where the folder "schematic" resides. So if your "schematic" sub-directory is say in the "lab2" directory, you need to be in the directory level that contains the "lab2" sub-directory.

2. Starting Quicksim:
   - In Unix, %quicksim *designname*
     Note: This assumes you are in same directory as your *designname*
   - Once quicksim has launched, maximize the screen. Clicking on the "large block" in your upper right hand screen.
3. Quitting Quicksim:
   At the "QuickSim II", select MGC > Exit

OPENING A SCHEMATIC DESIGN

- In the "Setup" window of Quicksim II, select the "OPEN SHEET" icon.
- After selection, your schematic should be displayed.
- Maximize the screen. Clicking on the "large block" in your upper right hand screen.

SIMULATING YOUR DESIGN

Discussion: First, you need to specify the input signals. Mentor Graphics calls this a *force*. You need to specify which signal is to be ``forced." You do this by selecting the signal, on your schematic, with your mouse. Simply click on the `A` port in your schematic. You should see the wire turn white denoting that it is selected. Reference (a). Some force commands can be found in reference (b).

1. Using a Force File (Recommended Approach)
   - Select signals (to trace on your design sheet)
   - Select "TRACE" in "Stimulus" window
   - In the "TRACE" window, right click the mouse
   - In the "QuickSim" pop-up window, select
     Force > From File Enter the ***forcefilename***

2. Manually Forcing an Input (steps)
   - At the "Setup" window in QuickSim II, select the "STIMULUS" button.
   - On the schematic, select the input desiring to "force". Note that the input path will be dashed and in white.
   - In the "Stimulus" window, select the "ADD FORCE" icon
   - Input the Value & Time pairs. Press OK.
   - Repeat with the next input signal (if any)

3. Displaying the Trace Window
   - Select all signals desiring to be traced. (using left mouse click)
   - At the "Setup" window in QuickSim II, select the "TRACE" button.

4. Opening a List Window
   - At the “Setup” window in QuickSim II, select the “LIST” button.

5. Running the Simulation
   - Click on the “List” window
   - Start typing “run xx” (where xx is the number of nanoseconds to be simulated).
   - Hit return
   - The “Trace” and “List” windows will display the results of the simulation.

6. Clearing the Simulation
   - in the “Design Changes Window”, select “reset” and then select “state” only. Enter OK.

7. Print Out Simulation
   - Select the desired window to be printed.
   - Select File > Print. Enter mentor_c207 as the printer name.

Comments:

- In the “Stimulus” window of QuickSim II, it provides you with the capabilities to “ADD CLOCK” and “FORCE TO STATE” as additional icons that can be used in the simulation.
- A print work-around is to “screen capture” your results. This is done by pressing “shift-PrtScn”. Then using MS Paint or MS Word applications, the screen capture can be “pasted” into the application. Finally, using the application print command, the simulation can be printed from Windows.