Lab 1 Schematic Capture

In the EE 552 labs, you will be using the Mentor Graphics and Actel Designer CAD tools. The Electrical and Computer Engineering department has 50 floating licenses for the Mentor Graphics tools set and six floating licenses for the Actel Designer software for the Sun workstations in CEB 531. The Mentor Graphics tools will also run on the HP workstations in CEB 540.

Labs should be submitted to the EE 552 assignment box outside of CEB 238 by 4 pm on the due date.

Lab reports should include:
A brief written summary of the results and how you arrived at them
Legible print-outs of schematics, code, simulations, etc. with titles and significant labels added (each page has to explain its self to the marker)

In this lab you will create hierarchical schematics for a 4-bit adder and a 4-bit subtractor using the Mentor Graphics tool set. Design Architect is the tool that you will be using for schematic entry. The following lab provides a step by step procedure for creating the schematic diagrams, and a symbol representation of the schematics.

In this lab you will:

Part 1 Enter a schematic

1. Open a new schematic sheet called full_adder.
2. Create a border.
3. Add symbol instances and create nets.
4. Check and save the sheet.
5. Use the selection filters.

Part 2 Create a symbol

1. Open a symbol editor window.
2. Create a symbol body called full_adder.
3. Add pins and properties to the symbol.
4. Check and save the symbol.

Part 3 Create a hierarchical symbol.

1. Open a symbol editor window.

2 Create a symbol body called add4.

3 Add pins and properties to the symbol.

4. Check and save symbol.

Part 4 Create a hierarchical schematic of a 4-bit ripple-carry adder

1. Open a new schematic sheet called add4.

2 Add symbol instances and create nets.

3. Check and save sheet.

Part 5 Design and create a hierarchical 4-bit ripple subtractor

PART 1

1. For this lab you can create two directories called `~/ee552` and `~/ee552/lab1`. You should move into the lab1 directory and at the command line type,

dmgr&

It will take a minute or so to bring up the Design Manager

2. Activate the Tools window and invoke Design Architect. (As a short-cut, you can alternatively start Design Architect from the UNIX prompt by typing

da&

a. Click the Select mouse button (LMB) on the Tools window.

b. To open the Design Architect Session window, double click the Select mouse button on the Design Architect icon.

3. Open a new schematic sheet and add a border.

a. After the Design Architect Session window is displayed, move the mouse to the palette area and click with the LMB on the Open Sheet icon.

The Open Sheet dialog box appears in the client area.

b. Fill in the Open Sheet dialog box with the following information:

Component Name: `~/ee552/lab1/full_adder`

c. Click on the Options button on the open sheet dialog box.

d. Click on the New Sheet button. This opens up additional options in the dialog box. In this lab we will use the default names “schematic” and “sheet1”, so do not change these entries.

e. If you click the Yes button for Sheet Borders and the Set button a Sheet Borders dialog box will appear. In this dialog box you can select the size of the drawing. A size A will be as shown in the Lab Schematic diagrams with full sized symbols on the first drawing and half size on the second. Do not include a Title Block unless you have a size D drawing or larger. (At this stage I don't know how to reduce the size of the Title Block).

f. Click the OK button on the Sheet Border Settings box.

The Sheet Borders dialog box disappears from the screen.

g. Click the OK button on the Open Sheet Option dialog box.

The Open Sheet Option dialog box disappears from the screen.

i. Click the OK button on the Open Sheet dialog box.

The Open Sheet dialog box disappears from the screen.

h. A sheet border has been added to the newly created sheet.

4. Practice using the Zoom and View functionality to view the sheet. For this you should practice using the View pull down menu. You can also use the Context window, where if you press the RMB a pop-up window will appear.

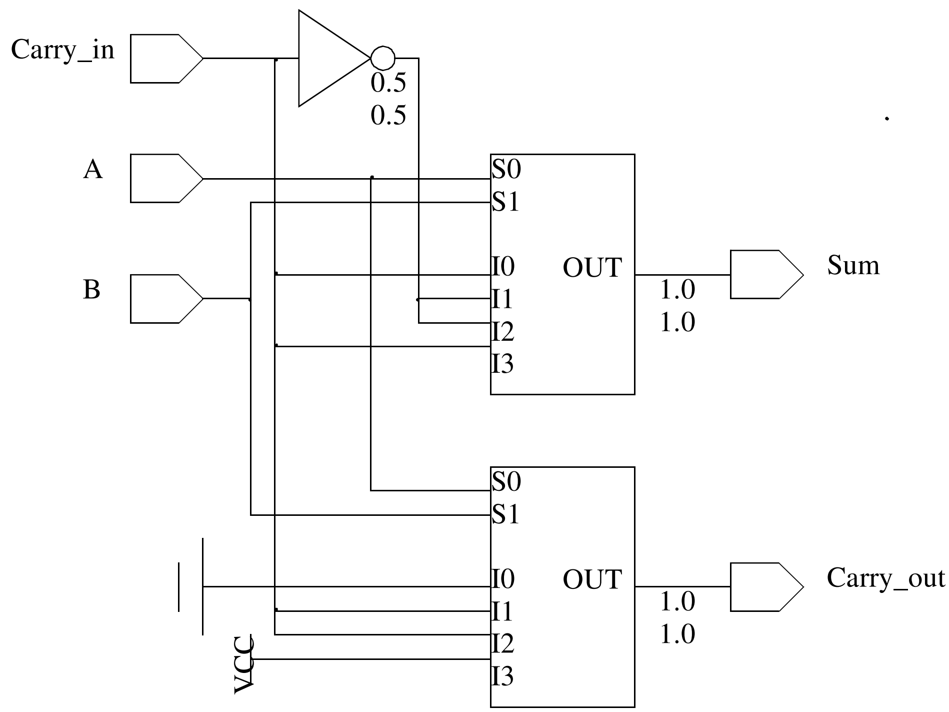

figure 1. Full adder

5. Add Instances to the schematic. You should create the schematic shown in Figure 1 using instances from the gen_lib (generic library).

a. Click on the Library icon on the palette window. This will bring up a list of MGC libraries that are available.

b Move to an open space on the library window and pressing the RMB use the pop-up menu to add scroll bars to the window.

c. Scroll down until you locate the gen_lib and select it. Again add scroll bars to this new window which lists all the parts in the gen_lib.

d. Click the mouse on the **mux41** component. Move the cursor into the edit area and you will see a ghost image of the **mux41** symbol. Click the LMB when the image is in the desired position.

e. Add a second instance of the **mux41** component.

f. Add an instance of the **inv** component.

g. Add an instance of the **ground** component.

h. Add an instance of the **vcc** component.

i. Select the ground instance and rotate it into position by using the pop-up menu

(Session) > Instance > Rotate/Flip > Rotate > -90

6. Add **portin** and **portout** symbols.

Using the same procedure as previous, add instances of the **portin** and **portout** symbols to the schematic.

7. Moving objects. When a component is selected it shows up as a white dotted outline. When it is not selected it has a solid color. If you press the LMB while it is pointing at a symbol you will 'toggle' the selection, i.e. if it was 'unselected' it will become 'selected', and visa versa. You can unselect all symbols by using the Function Key F2.

a. If we select one instance we can use the pop-up menu in the Session Window to move selected items,

(Session) > Instance > move > selected

A ghost image of the selected items can be moved to the desired location.

8. Add nets

a. Press the F3 (Add Wire) function key.

The Add Wire prompt bar appears in the Schematic Editor window.

b. Move the cursor to an instance pin vertex. Click the Select mouse button at the desired location of the net's initial vertex. Move the mouse to the next net vertex. If this is the terminal vertex double click the Select mouse button. If it is an intermediate point, single click the Select mouse button.

c. Using this procedure complete all the nets on this sheet.

d. When you have completed adding nets, click the Cancel button on the still-displayed Add Wire prompt bar.

e. Press the F2 (Unselect All) function key.

9. Change Net property values.

Note that every instance of a **portin** or **portout** has the name NET associated with it. Each of these must be replaced by a unique name. If you select one **portin** symbol and press the RMB a pop-up menu will appear. If we select

(Session) > Instance > Properties > Change Text Values

The dialog box that appears has a change property value box. If we type `Carry_in` and hit the TAB key a second box appears where we can type in `A`, then `B`, `Sum`, `Carry_out`, and finally the click the button OK. After we have typed in all the **portin** and **portout** names a line then appears at the bottom of the Session Window saying which name will be placed on the next selection. If we click on a NET name the name `Carry_in` will replace that NET name. We then continue with A, then `B`, `Sum` and `Carry_out`.

10. Check the sheet

From the menu bar select the following menu items

Check > Sheet > With Defaults

A Check Sheet Results window will then be displayed.

There should be no errors and no warnings in this example.

11. Save the Sheet

From the menu bar choose the following item

File > Save Sheet

12. Select Property Text

By default property text is not selectable. In order to select text information click on the Set Select Filter on the palette. A dialog box appears which you change to Set All. Then click OK. Now the rise and fall times can be selected. Select the rise and fall times and change them to 0.5 for the inv and 1.0 for the muxxes. (Alternatively you can first clear all the entries and select Property entry only.)

13. Click on the Set Select Filter on the palette menu and reset the filter to the MGC Defaults.

14. First check and then Save the sheet.

PART 2

1. Return to the Design Architect Session window by closing the sheet1 schematic of the `full_adder`.

2. Open a new Symbol Editor window.

a. From the palette menu of Design Architect select the Open Symbol icon. A dialog box will appear, fill in the following information:

`~/ee552/lab1/full_adder`

b. Click the OK button

3. Create a symbol body

a. Use the palette menu to select Rectangle to create a rectangular symbol for the full adder.

b. By using F2 to unselect features you can use the mouse to select a single feature. You can then use the pop-up menu to

> Move

> Copy

> Delete > Selected

the selected item.

4. Add pins to the symbol body

a. Click on the Add Pin icon in the "symbol_draw palette" window.

b. In the dialog box that appears use the following entries:

Name Placement: Manual

Pintype: IN

Pin Placement Left

Pin Names: `Carry_in`

A

`B`

c. Click the OK button.

d. A prompt bar appears, move the cursor to the desired position and click the Select mouse button. Next move the cursor to position the text in the desired position.

e Continue with the A and B pins

f. Select the Add Pins icon again but this time make the pins outputs with a Right orientation. These two pins should be named `Carry_out` and `Sum`.

5. Check the Symbol

a. Using the defaults for checking levels, check the symbol for any errors. From the menu bar select the following item:

Check > With Defaults

b. If you have no errors close the Check Status window.

6. Save the Symbol

From the menu bar select the following item:

File > Save Symbol

7. Check the Symbol again

PART 3

Create a symbol diagram with the signals `Carry_in, A0, A1, A2, A3, B0, B1, B2, B3, Sum0, Sum1, Sum2, Sum3, Carry_out`, run a check on the symbol and save the symbol. (with default registration).

While the symbol session is still active use the menu bar to select

(Menu) > Miscellaneous > Create Sheet...

Fill in the dialog box, click OK and a new Design Architect sheet will open with the **portin** and **portout** icons with appropriate NET names already placed.

PART 4

Create a schematic of a 4-bit ripple-carry adder using the 1-bit full adders. Keep these files around for the next lab.

What is the worst case delay through the 4-bit adder?

PART 5

Create a hierarchical design of a 4-bit subtractor as you did for the adder. The 1-bit full subtractor should have the signal names `A, B, Borrow_in, Difference` and `Borrow_out`. The `Difference` is `A` minus `B` minus `Borrow_in`. First draw a truth table for the 1-bit full subtractor. Create a schematic using only the components **mux41**, **mux21**, **inv** and ports. Try to minimize the number of components.

Hint: opening the adder schematics, saving them under a new name and making changes may save a little time.

Hand in hard copies of the appropriate schematics for this lab and answer all questions.