0 Introduction

- You may want to review this chapter as you progress through the tutorial. Some of the concepts and terms may not be familiar until later chapters.

0.1 Overview of Object Oriented Data

Mentor Graphics is not a single tool. It is a series of design tools that uses object oriented data to simplify a complete design process. What this means is that data created in one tool can be shipped to another tool for further processing. At this point an example is in order.

A schematic is merely a pictorial representation of a circuit. When a schematic is created in Design Architect, the data is saved in two objects: a component and a viewpoint (See Figure 0.1). A component is a collection of models used to describe the functional, graphical, and other aspects of the design. A viewpoint can be thought of as the filter that an application uses to process component data.

In order to analyze the circuit, the component data is passed to the simulator, know as QuickSim. QuickSim creates another, different viewpoint for the data. Local changes may be made to the schematic without changing the actual data from Design Architect.

In summary, a Mentor Graphics design object is a collection of a component object and various viewpoints. Applications use a viewpoint to operate on component data.

0.2 Overview of Component Data

For most of the designs you will be creating, the component data is made up of a schematic and a symbol. A symbol is a graphical model of the input and output pins. A schematic is a functional model of how output values are related to input values.

Please be aware that these are tightly coupled entities. If you make a schematic named “80486” in one directory and a symbol named “80486” in another directory, they will not be associated together. If you later use that symbol in another schematic, the simulation will fail. To avoid this problem, be sure that you are opening a sheet /symbol on the same design data object.

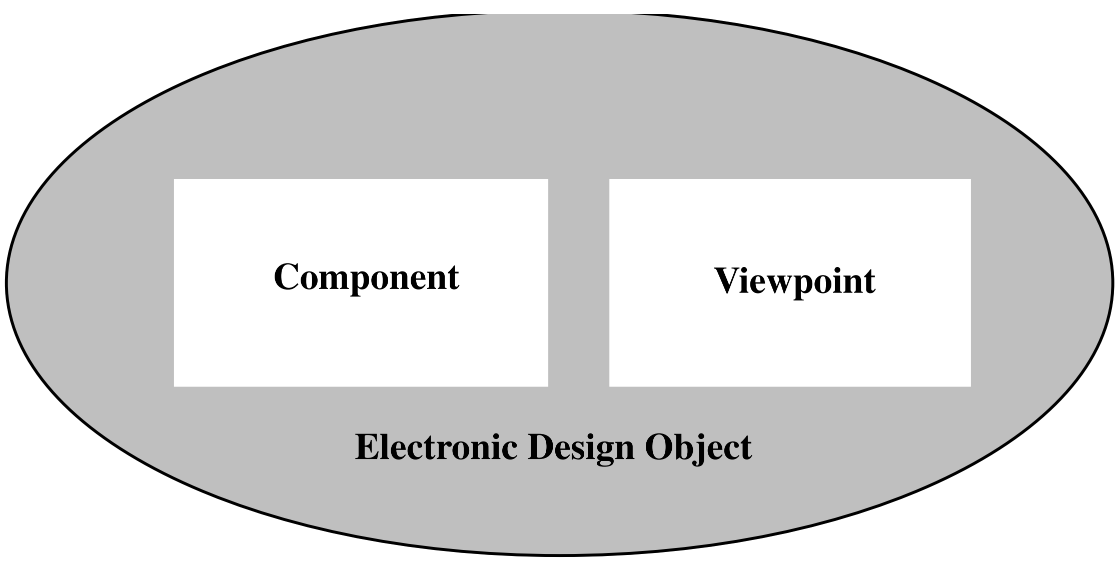


Figure 0.1

Figure 0.2 shows the hierarchy of a design object to implement the logical function XOR.

0.3 Moving Design Data

For those students familiar with the UNIX operating system, refrain from using UNIX commands to move directories or designs. You MUST move these objects using the Design Manager. Failure to use the Design Manager will result in data corruption and possible loss of designs.

Even if you use the Design Manager to move data, there are potential problems. Many of the designs you will do this semester will have a symbol associated with the schematic. The symbol will then be used as a building block in other designs. When you place your symbols in other schematics, Design Architect stores the absolute pathname to the design. If the design is moved, then QuickSim will not be able to find the component data for that symbol and may lock-up. The real problem is that the schematic with the invalid symbol will still check correctly. One method to check for this problem is to select the symbol and perform *(Schematic Popup) Open Down...* If there is no entry for a schematic, then the symbol is invalid. To correct this, delete the symbol and replace it using *Choose Symbol*.

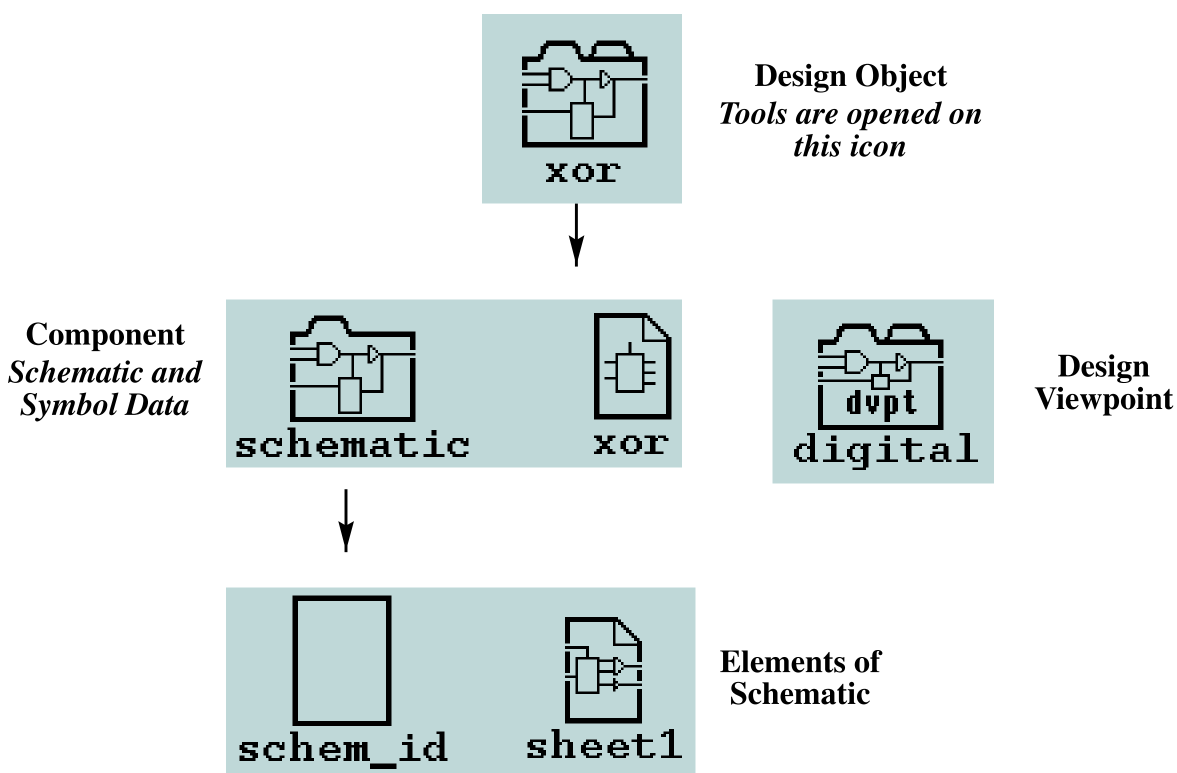


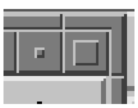
Figure 0.2

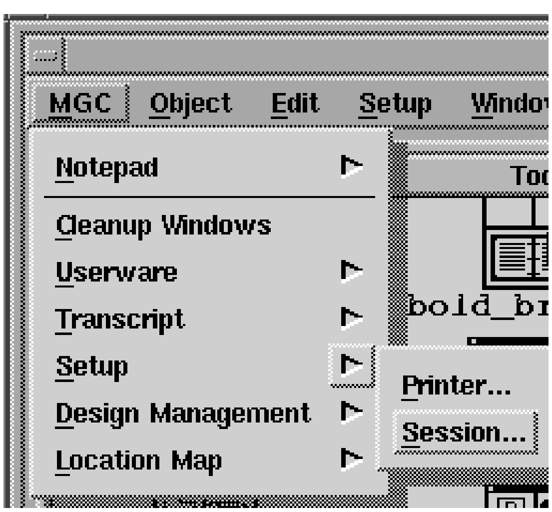
1 Falcon Framework

1.1 Getting Started

- ***GSFF Section 2; 4-5 to 4-8***

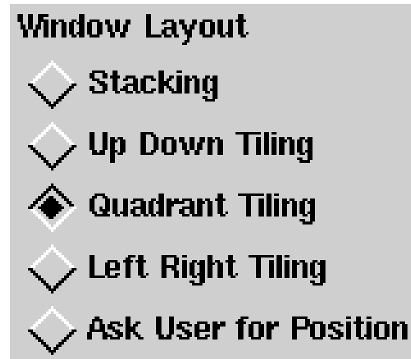
After logging into a workstation, various windows will appear. Type **ecec03** Execute this script every time you wish to do work for your class. After executing the script, your mentor working directory will be set to *<random_chars>/MGC/ecec03*. Type **echo $MGC_WD** from the UNIX prompt after you execute the startup script to see the exact value.

From a UNIX prompt, type *dmgr*. After a moment, a Design Manager window will appear. The size of the screen is probably too small to see all of the menus. Press the square in the upper right corner of the outermost window . The Design Manager window will fill the entire screen. You will now see three windows, to the left is the Tools Window, on the right is the Pallete Window, and in the middle is the Navigator Window. If you click on the same square again, the window will return to its original size. Execute the following:



(Menu bar) MGC > Setup > Session...

After a moment, a dialog box will appear. Click on



to allow up to four windows to be displayed simultaneously. Then click *OK* to save the changes.

The MGC pulldown menu is available at any time in any application.

1.2 Prepare for a Design

- ***GSFF 4-5 to 4-8***

Click the *select* (left) mouse button on the *Navigator* window. (The *Navigator* window is the window to the right of the *Tools* window that says *<random_chars>/MGC/ecec03*. Do not be confused. The long window next to the *Navigator* window is a palette. It will not be used in this chapter.) The *select* mouse button is always used to activate a window. After a window is activated, the outline should change color and the menu bar should show several options. Click the *select* mouse button on the *Tools* window and notice how the *Add* option disappears. Click the *select* mouse button on the *Navigator* window (The *Add* option will appear again). All Mentor commands are dependent upon which window is highlighted or **active**. If a particular command menu is not available, then it is highly probable that the wrong window is active.

Every new lab that you do should have a separate directory.
Execute *(Menu bar) Add > Directory:*. A

prompt bar will appear. Enter **lab1** and click *OK*. A directory icon will appear. Double click on the *tutorial* icon. The *Navigator* title area will read *<random_chars>/MGC/ecec03/lab1*. Spend a few moments using the navigator buttons to learn how to traverse the UNIX file system. Return to the lab1 directory.

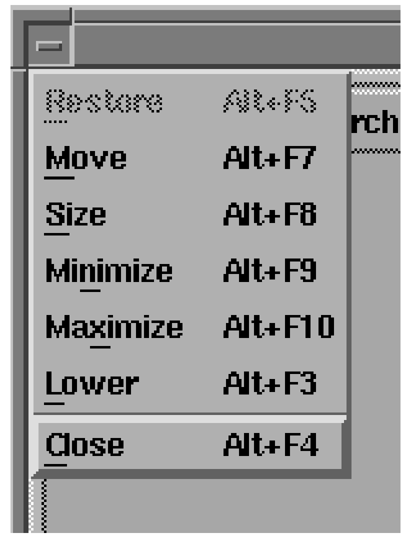
1.3 Bold_Browser

- ***GSFF 6-6;6-10***

Move to the *Tools* window and click to activate. Using the scroll bars, locate and double click the *bold_browser* icon. Click twice on *Falcon Framework.* Use the scroll bars to find and click on the *Common User Interface Manual.* Click *OK.* Read the section on strokes. Strokes will save you a lot of time, but you will have to use the *bold_browser* to find out more. Spend some time examining the on-line documentation. The entire Mentor Graphics documentation library is contained within the BoldBrowser.

1.4 Exiting Mentor Applications

All Mentor Graphics applications are terminated the same way. Use the following to close *Bold_browser* (but do not close *dmgr* at this time).



(Outer Window Pulldown) Close

1.5 Murphy's Computer Law

If builders built buildings the way programmers wrote programs...
the first woodpecker that came along would destroy civilization.

2 Basic Schematic Capture

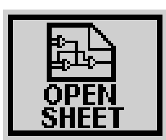
2.1 Start Design Architect

- ***DATW 2-10 to 2-11;2-8 to 2-9;2-52 to 2-53***

The circuit to be constructed is an XOR gate. The circuit appears in Figure 2.1.

Design Architect schematic windows have two options for popup menus: Selection Free and Selection Sensitive. Selection Sensitive Menus change when objects within a window are selected. Selection Free Menus are the same regardless of what objects are selected within a window.

To invoke Design Architect, double-click on the *design_arch* icon in the *Tool* window. You should enlarge the Design Architect window to see all of the menus. When the Design Architect window appears, do the following.

- Click 

on the *session_palette.*
- Enter **<random_chars>/ecec03/lab1/my_xor** next to *Component Name.* Click *OK.*

CAVEATS:

- When you open a sheet, you will see a box for a sheet name. Changing the sheet name does NOT mean you are starting a new design. A new sheet is for adding extra pages to a design. So if you are required to create different versions of a latch, for example, be sure to use different *Component Names***.** You will also have to create a new symbol for each design.
- When naming components, do NOT use spaces. Design Architect will accept them. QuickSim II will not.
- No software is perfect. Mentor Graphics is no different. If Design Architect crashes, you may have to delete some lock files. Use the appropriate UNIX commands to find and delete all files ending with ".lck" in your design directory. You may have to go down several directories to find them all.

2.2 Place A Border and Comment Block

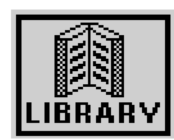
The purpose of the border is to provide a boundary of legibility. The comment block is for name, design, etc. To include a border and comment block, execute *(Schematic Menu Bar) Edit >Add Sheet Border*

Enter the appropriate information in the boxes. Since the comment block is used by many classes, some information may not be necessary. You should check with your TA to be sure.

2.3 Place Parts

- ***DATW 2-16 to 2-17 ; 2-38 to 2-39***

Execute the following commands to place the necessary components.

- Execute *(SchematicMenuBar)Libraries > MGC Digital Libraries*
- Execute *(SchematicMenuBar)Libraries > Set Default Palette*
- Now you can click 

on the *schematic_palette* to get to the MGC Digital Libraries Palette
- Click on *gen_lib.*
- Use the **<PageUp>** and **<PageDown>** keys to locate *nand2*. Click on *nand2* and move the ghost image onto the schematic. Click the *select* mouse button to place the symbol. Place three more *nand2* components.
- Locate and place two *portin* components and one *portout*. The schematic should look like Figure 2.2.

2.4 Wire Nets

- ***DATW 2-24 to 2-28 ; 2-46 to 2-47***

Using Figure 2.3, connect the appropriate pins and nets together. Execute *(Schematic Pop-up)Schematic>Wire:*. A tan popup dialog box will appear in the lower left hand corner. Move the cursor to the first point and click the *select* mouse button. Move the cursor to the second point and click *select* again. You have just created a net segment. Each time you click *select*, the net will be extended to that point. To start another net, press **<Enter>**. When you have finished wiring, press the **<ESC>** to cancel the popup box. Save your work using *(Schematic Menu Bar)File > Save Sheet > Default Registration*. Saving early and often is a wise idea.

CAVEATS:

- When starting subsequent nets, be sure to press *<CR>* or double-click the *select* mouse button to begin a new net.
- Notice that the prompt bar may not disappear. The same prompt bar may be used to wire every net in the design. It will not go away until 

or **<ESC>** is pressed.
- Be sure to click on the last point to be connected before pressing *<CR>*. Otherwise, the last point will not be connected.

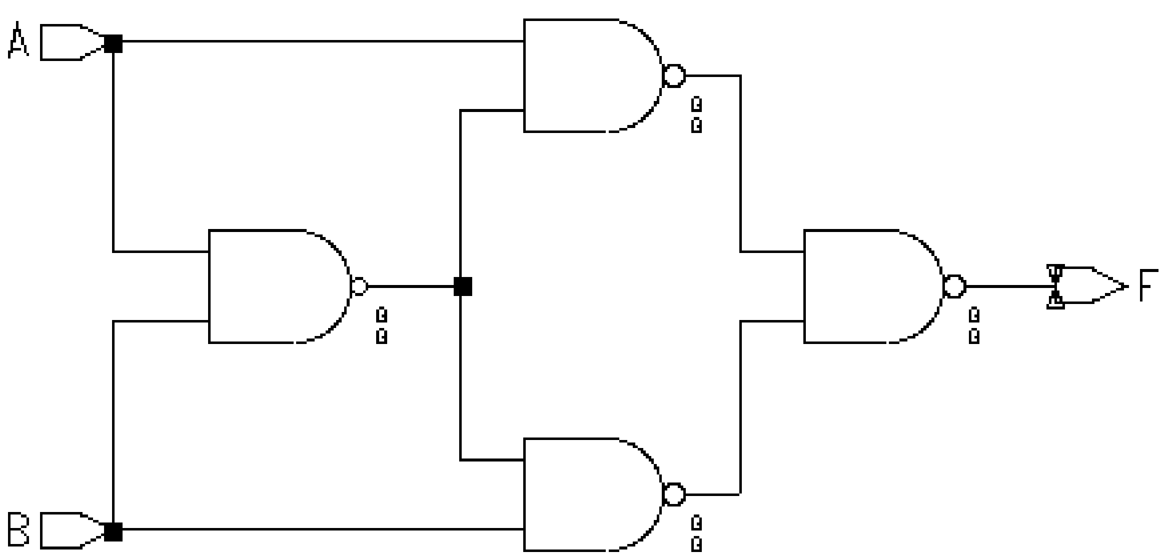


Figure 2.1

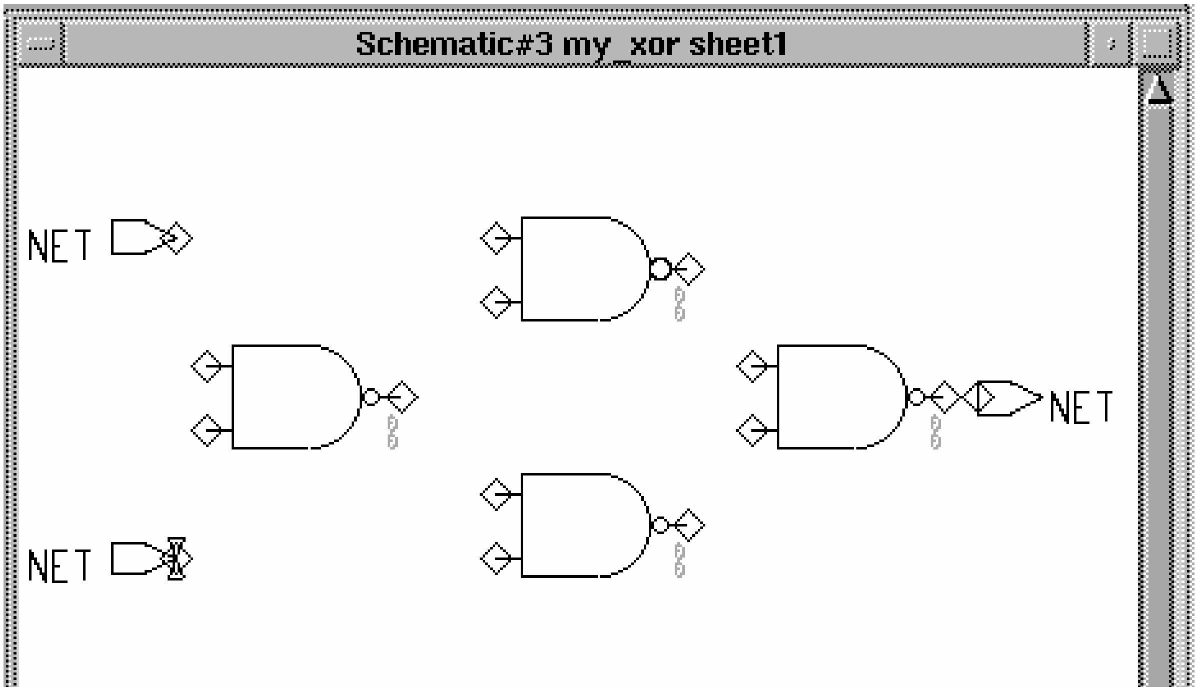


Figure 2.2

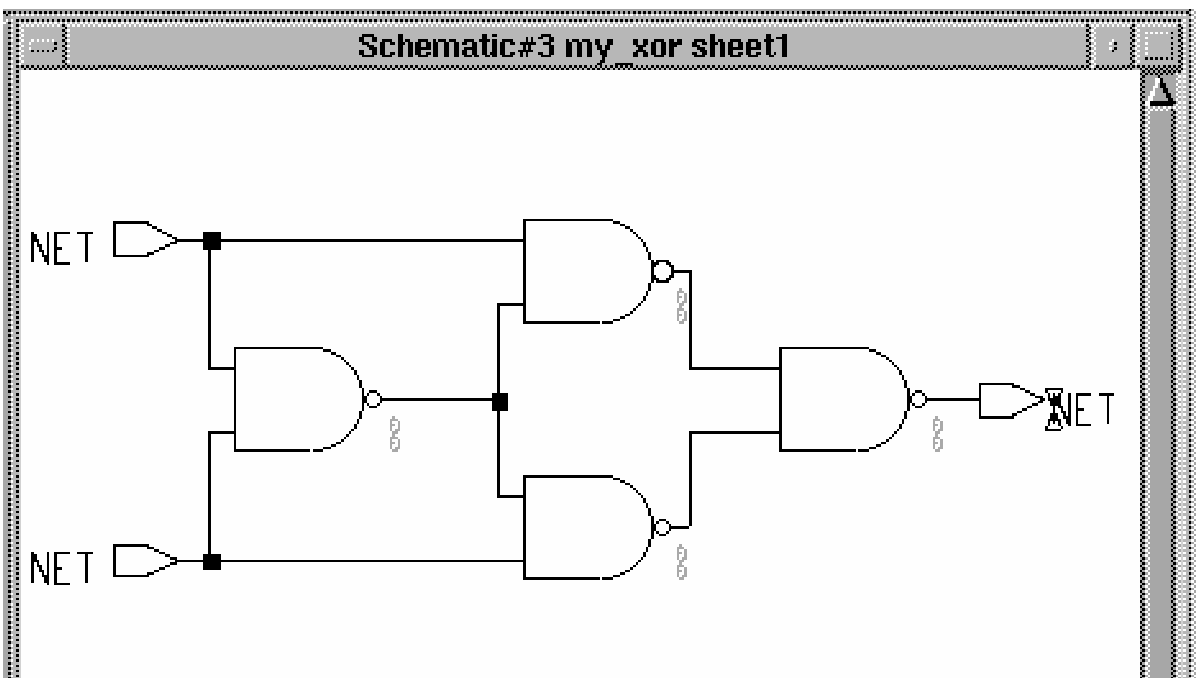


Figure 2.3

2.5 Changing *NET* Properties

At this time, all of the connectors have the text *NET* next to them. As indicated in Figure 2.4, the labels need to be changed to *A,B,* and *F* respectively.

- Use the key sequence **<shift><F7>** with your cursor directly over the word *NET* to change the value (This will work for any property or piece of text). Enter the appropriate value in the *New Value* box.

Save your work one more time. At this point your schematic should closely resemble Figure 2.4.

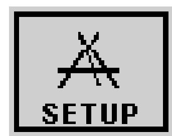
2.6 Perform Design Rule Checks

Execute *(Schematic Menu Bar) Check > Sheet > With Defaults.* A window will appear that lists possible design violations. Correct any violations and re-run the check. Once the check returns with no errors and no warnings, save the design.

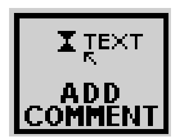
2.7 Add/Change Comments

- ***DATW 4-8 to 4-11***

If you wish to add additional comments, press the TEXT box in the *schematic_palette*. You may need to execute *(Schematic Palette Popup)>Schematic Palette>Display Schematic Palette* to return to the root palette.

- 

changes the display/printing parameters of text. A font height of 0.250 is reasonable for most printouts. Be sure that the comment button, NOT the property button, is pressed.

- 

adds comment text to your schematic.

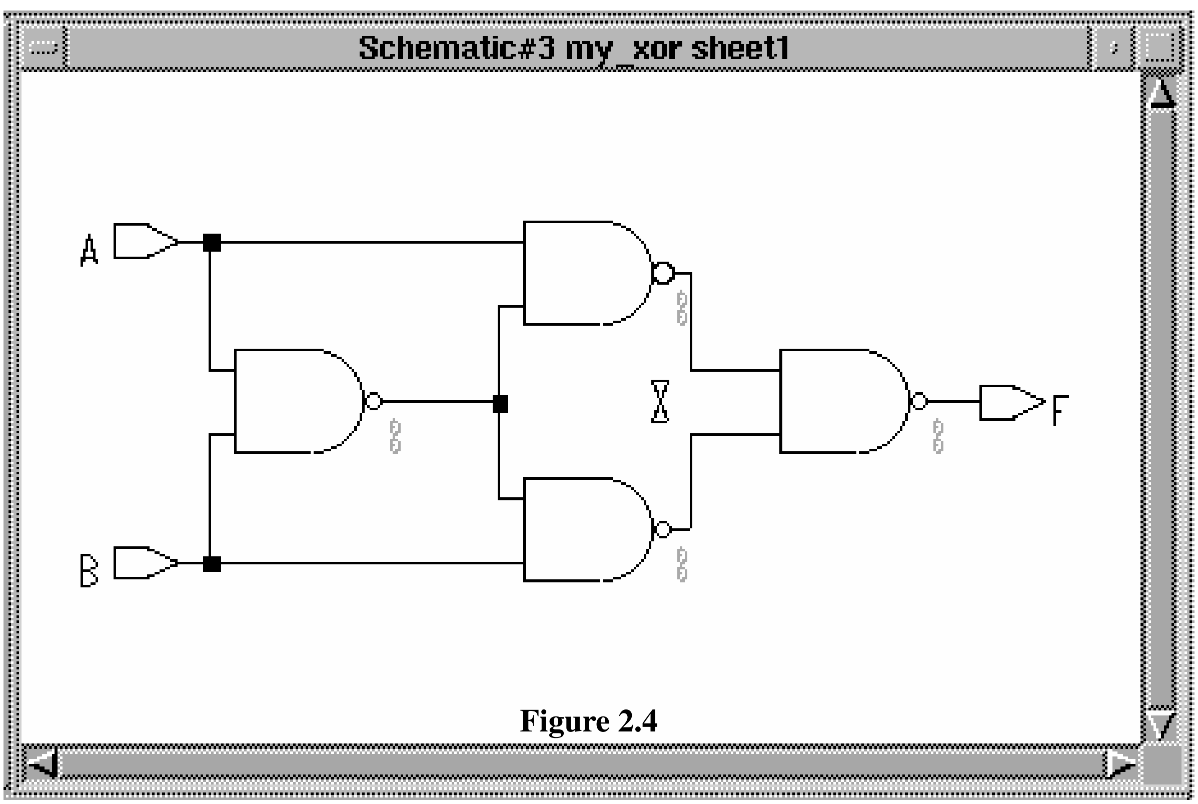


Figure 2.4

- By pressing 

, text or any other object can be interactively placed. But be sure it is highlighted in a red box. Only sections with the red box are active.
- The sequence **<shift><F7>** with your cursor directly over any text or property will change a value

After comments are added and the design is checked, save your work.

2.8 Printout Schematic

To print out the schematic, execute *(Schematic Menu Bar) File > Print Sheet:*. Click *OK*. Leave the schematic window open for a moment, but be sure to save your work.

2.9 Murphy's Computer Law

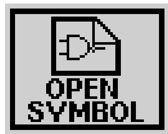
Every task takes twice as long as you think it will take. If you double the time you think it will take, it will actually take four times as long

3 Creating Symbols

3.1 Open a Symbol

- ***DATW 3-4 to 3-5***

If necessary, click 

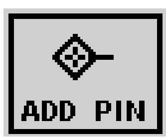
on the *schematic_palette.* Click 

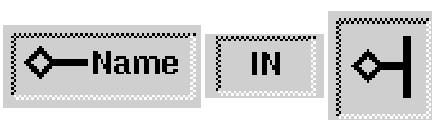
. Enter the EXACT name you used to open the sheet or click on the *Navigator...* button to find the **my_xor** design object. Do NOT add a border or comment block for symbols.

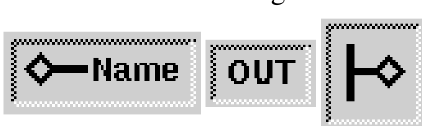
3.2 Add Pins and Perform Checks

- ***DATW 3-14 to 3-15***

Perform the following steps to add and to name the pins for the circuit my_xor.

- Press 

.
- Make sure the following buttons are pressed 

.
- Enter **A** in the *Pin Name(s)* box. Notice that a new, blank box appears every time text is typed into a box. Enter **B** in the new box that appears. Click *OK.*
- Using Figure 3.1 as a reference, move the cross hairs and click the *select* mouse button to place the two pins.
- Press the *ADD PIN* button again. This time, make sure the following buttons are selected. 

.
- Enter **F** in the *Pin Name(s)* box. Place the final pin in accordance with Figure 3.1.

3.3 Create a Symbol Body

- ***DATW 3-10 to 3-11***

Perform the following steps to create a symbol to represent the my_xor circuit.

- *(Symbol Menu Bar) Setup > Symbol Body...*
- Set the *Line Width* to *3 pixels* click *OK.*
- *(Symbol Popup)* ***SYMBOL*** *> Add Graphics > Rectangle:.*
- Place the cursor where you want one corner and drag until a suitable outline is achieved (about 4 large dots by 4 large dots is fine).
- Use the *ADD TEXT* button to add the comment *=1* to the symbol body.

The final result appears in Figure 3.1. Check and correct any problems with the symbol. (The first time you run a check you may get three warnings. If this occurs, save the symbol and re-check the design.) Finally save the symbol and close the symbol window.

3.4 The Easy Way

In the future, the command *(Schematic Menu Bar) Miscellaneous>Generate Symbol...* can be used from the schematic (not symbol) window to automatically generate a symbol.

3.5 Murphy's Computer Law

Any sufficiently advanced technology is indistinguishable from magic.

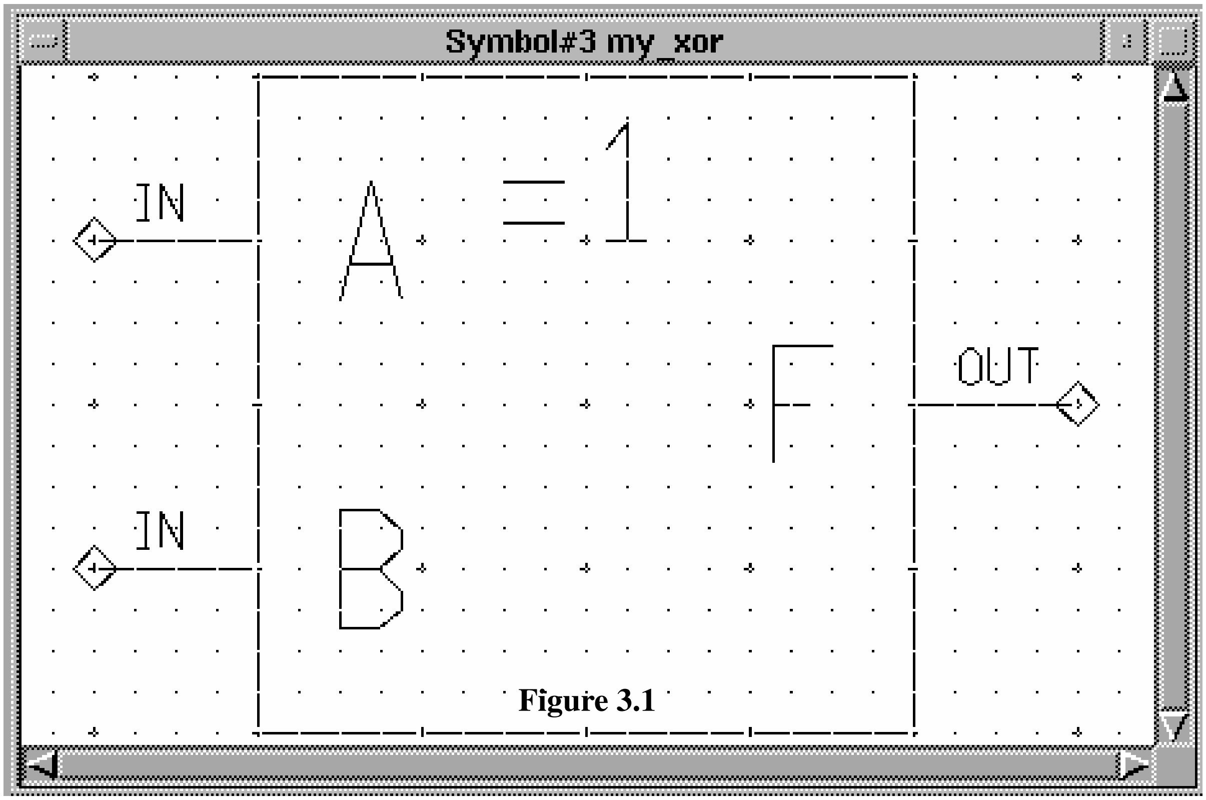


Figure 3.1

4 Capture II: Return to DA

4.1 Invoke Design Architect

The symbol from Section 3 will be used to create a full-adder. The adder will take two numbers plus a carry-in and generate a sum and carry-out. See Figure 4.1 for a schematic of a full-adder. Open a new sheet and use the component name **$<random_chars>/MGC/ecec03/tutorial/full_adder**. Refer to Section 2 for help on how to open a new sheet.

4.2 Place Standard Parts

Remember to add a border and comment block. Refer to Section 2.2 for help on how to do this. All schematics, but NOT symbols, should have a border and comment frame.

Using Figure 4.2 as a guide, place the *portin, portout,* and *and2* components. At this time, change the *NET* name property of the *portin* and *portout* components to *A,B,Cin,Sum,Cout.*

4.3 Place Custom XOR and Wire Nets

- ***DATW 2-16 : 2-17***

Move to the schematic window. Execute *(Schematic Popup)* ***SCHEMATIC*** *> Instance > Symbol By Path...* Use the navigator to select the my_xor design object created in Section 3. Place three instances as shown in Figure 4.2. Follow the schematic in Figure 4.1 to add the appropriate nets and to change the appropriate *NET* name properties. The final result should look like Figure 4.3. Check and save the design. The next step is to analyze the XOR gate and the full-adder.

4.4 Printout Schematic

Follow the directions in Section 2.8 for getting output. Do not forget to add comments to the schematics. Close the schematic window.

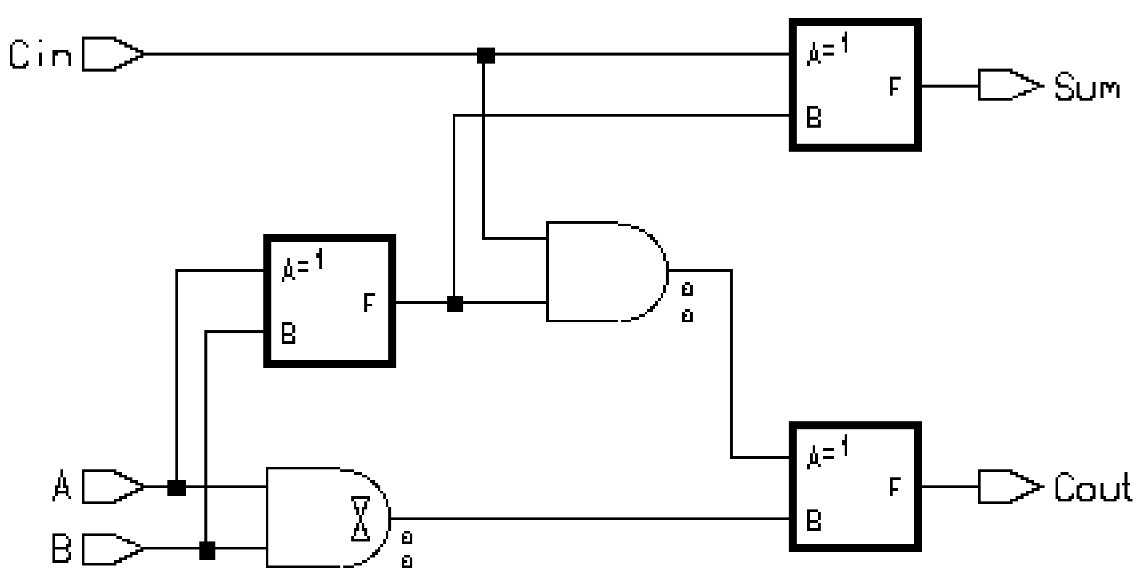


Figure 4.1

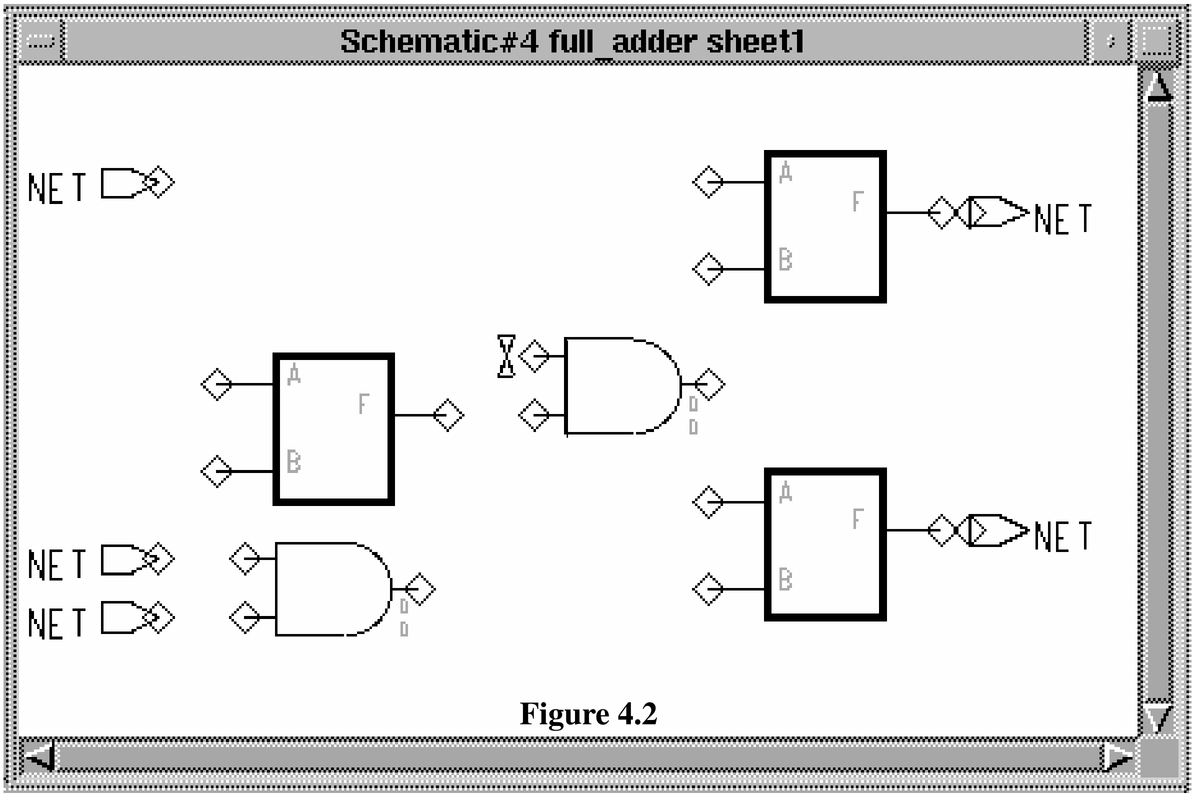


Figure 4.2

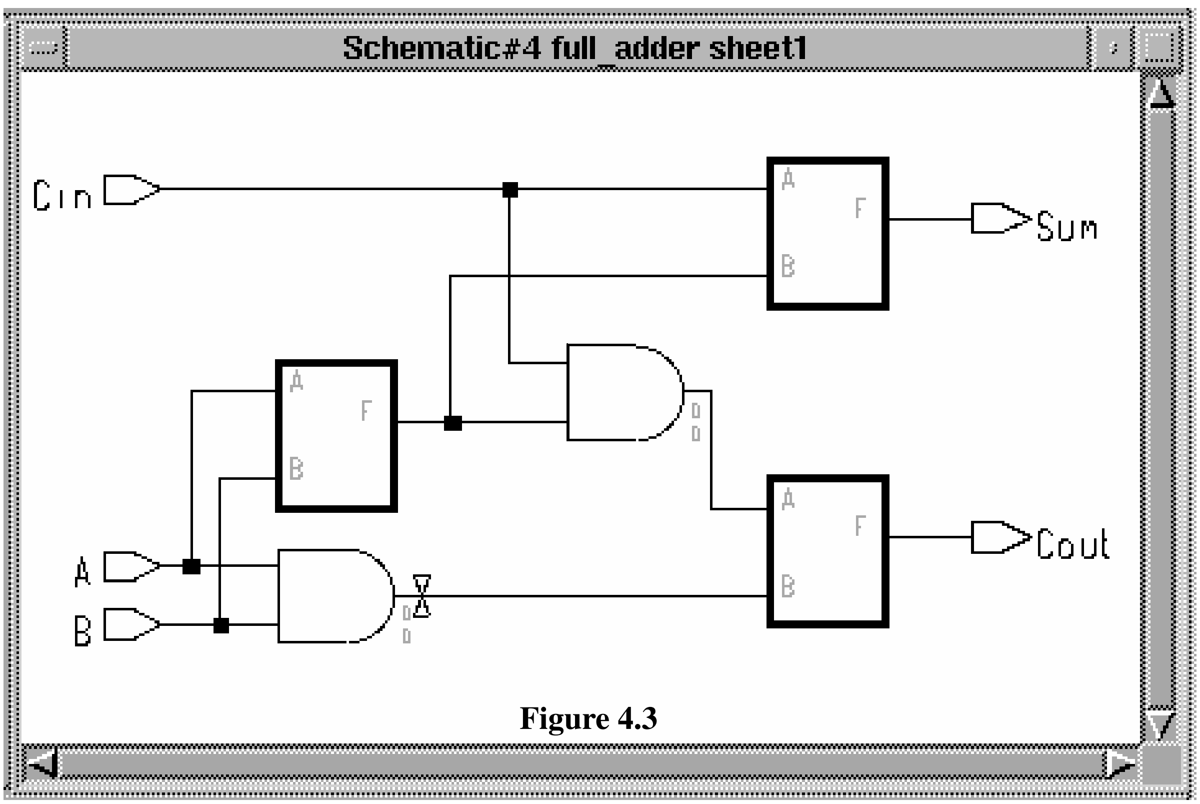


Figure 4.3

5 Logic Analysis

5.1 Invoke and Setup QuickSim II

- ***SVTW 2-8 to 2-9 ; 2-14 to 2-19***
- ***QSTW 1-12 to 1-13***

To invoke QuickSim, activate the *Navigator* window in Design Manager and execute *(Popup)* **Navigator** *> Update Window.* Click once on the *my_xor* icon. Then execute *(Popup)* **Navigator** *> Open > QuickSim II.* Another way QuickSim may be invoked is from the *Tools* window. Execute the following to setup the QuickSim environment.

- *(Menu Bar) Setup>Kernel>Analysis...* Click *Delay.* Click *Visible.* Click *OK.*
- *(Menu Bar) Add > Traces...*
- In the *Signal Name* box enter **A**, **B**, and **F**. Remember that each signal is input in a separate *Signal Name* box. Click *OK.*
- When the *Traces* window appears, click on each signal name.
- *(Menu Bar) Add > Lists...* This time 

is depressed. The signals that were highlighted are now automatically selected for inclusion in the *List* window. You can select as many or as few signals for each window and can use either method for signal selection.
- *(Menu Bar) File > Open Sheet*. The schematic is brought into the Quicksim environment. Notice that selected signals are highlighted on the schematic. Another method of selection is to select the signal off of the sheet, then add it to the *Trace* or *List* window.

5.2 Set Forces

- ***SVTW 4-8 to 4-11***

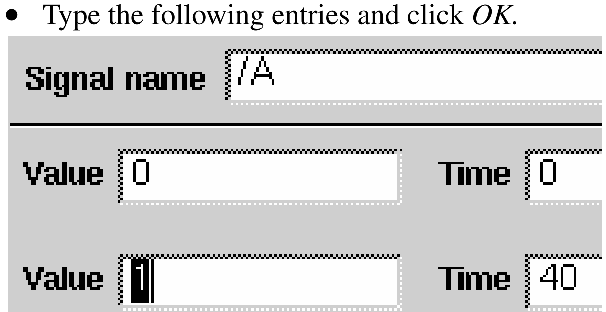
The simulator is now ready to analyze your circuit. However, the values of the input pins need to be defined before any output values can be determined. Table 5.1 is a truth table for the my_xor component.

A	B	F
0	0	0
0	1	1
1	0	1
1	1	0

Table 5.1

To test the circuit, the inputs *A* and *B* have to be set or **FORCED** to a specific value. Then the simulator will compute the output *F.* Execute the following to create the necessary forces.

- Select signal *A* only (make sure *B* and *F* are deselected).
- *(Trace Popup)* **QuickSim** *> Force > Multiple Values...*
- Type the following entries and click *OK.*



- Repeat for signal *B* with the following values.

Value	Time
0	0
1	20
0	40
1	60

5.3 Run the Simulation

- ***QSTW 1-18 to 1-21***

QuickSim is now ready to exercise the circuit. Execute *(Trace Popup)* ***QuickSim*** *> Run > Simulation > For Time:*. In the prompt bar, enter the value **80**. The results should look like Figure 5.1. If the results are incorrect, exit QuickSim and return to Design Architect to correct any problems.

5.4 Printout Results

The menu choices available for printouts vary according to which window is active. Usually, the *Trace* and *List* windows are the only results that will be required.

For the *Trace* window, the *(Trace Popup)* ***QuickSim*** *> View...* commands will be helpful for selecting portions of the waveforms to print. Execute *(Trace Popup)* ***QuickSim*** *> View > View All* for this tutorial.

For either window execute: *(Menu Bar) File > Print > Active Window...* Enter **0**, **100**, and **mentor** for *Begin Domain, End Domain,* and *Printer Name* respectively.

When you are finished, exit from QuickSim. Anytime you make a change to your schematic or want to analyze a different circuit, you have to exit and re-start Quicksim.

5.5 For Future Reference

As you may have noticed, QuickSim takes a long time to load. Exiting QuickSim and starting it again is frustrating and time consuming. But if you need to make changes to the top level schematic, there is no other way. However, if you need to make changes to the schematics underlying the symbols on the top level, you may use *(menu bar) File > Load > New Models > Selected* to re-load the changes you have made.

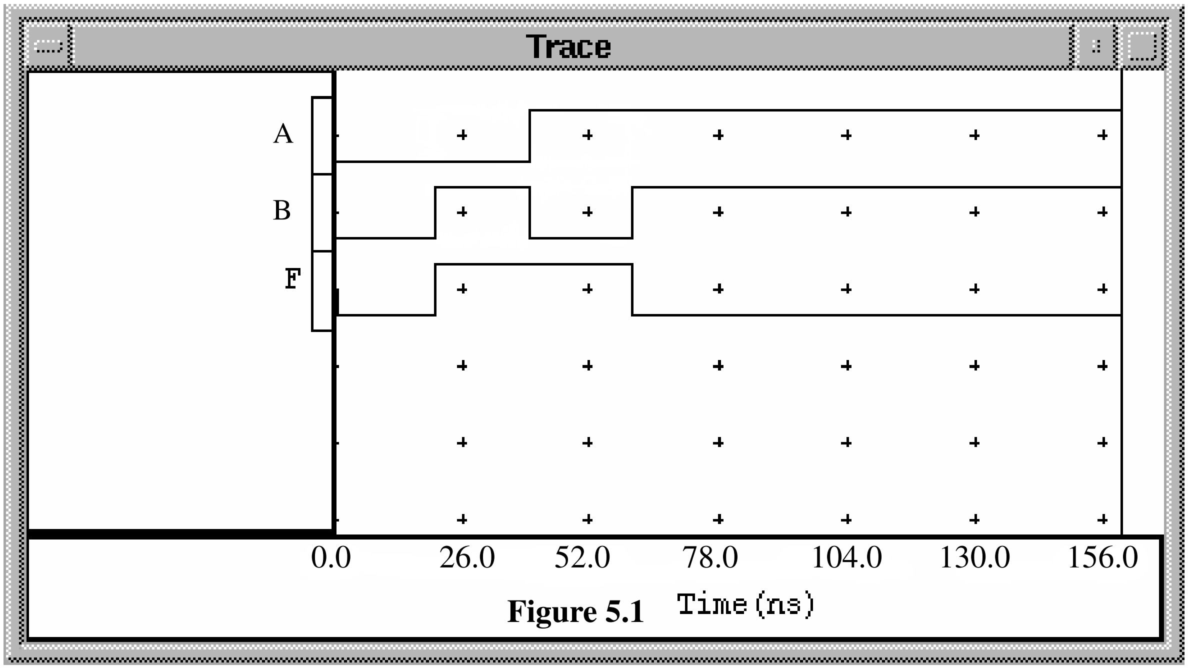


Figure 5.1

5.6 Simulate the Full-Adder

- ***QSTW 6-4 to 4-7***

Invoke QuickSim on your full-adder circuit.

The following truth table describes the functionality of the full-adder.

A	B	Cin	Sum	Cout
0	0	0	0	0
0	0	1	1	0
0	1	0	1	0
0	1	1	0	1
1	0	0	1	0
1	0	1	0	1
1	1	0	0	1
1	1	1	1	1

Table 5.2

As designs get more complex, re-entering forces is not practical. There are three methods by which forces can be created/saved for later use.

- Type the force commands into a file using **vi** or whatever editor is available. Then load them by typing *"DOF **<Pathname>/<Filename>**"* from anywhere in QuickSim. For an example file, look at ~cad/mgc/sample_dof.
- Create the force commands using *(Traces Popup) Force > Add...* The forces can then be saved as a forcefile or Waveform Database. A forcefile may be edited by hand if necessary.
- Create the force commands using the Waveform Editor. In this case, the forces must be saved as a waveform database.

The merits and disadvantages of these methods are covered in the **Mentor Reference Guide**. The **vi** editor is covered in the **College of Engineering Workstations Users Guide**.

The following tables contain the necessary force values and times to thoroughly exercise the full-adder circuit. Run the simulation for 80 time steps. The results should be similar to Figure 5.2.

Value	Time
0	0
1	10
0	20
1	30
0	40
1	50
0	60
1	70

Table 5.3 Forces for Cin

Value	Time
0	0
1	20
0	40
1	60

Table 5.4 Forces for B

Value	Time
0	0
1	40

Table 5.5 Forces for A

5.7 Printout Results

At this time, both schematics and the my_xor *List* and *Trace* windows should be printed. Print out the simulation results for the full_adder. Your instructor will announce what will be necessary for submission.

5.8 Murphy's Computer Law

If all else fails, read the directions.

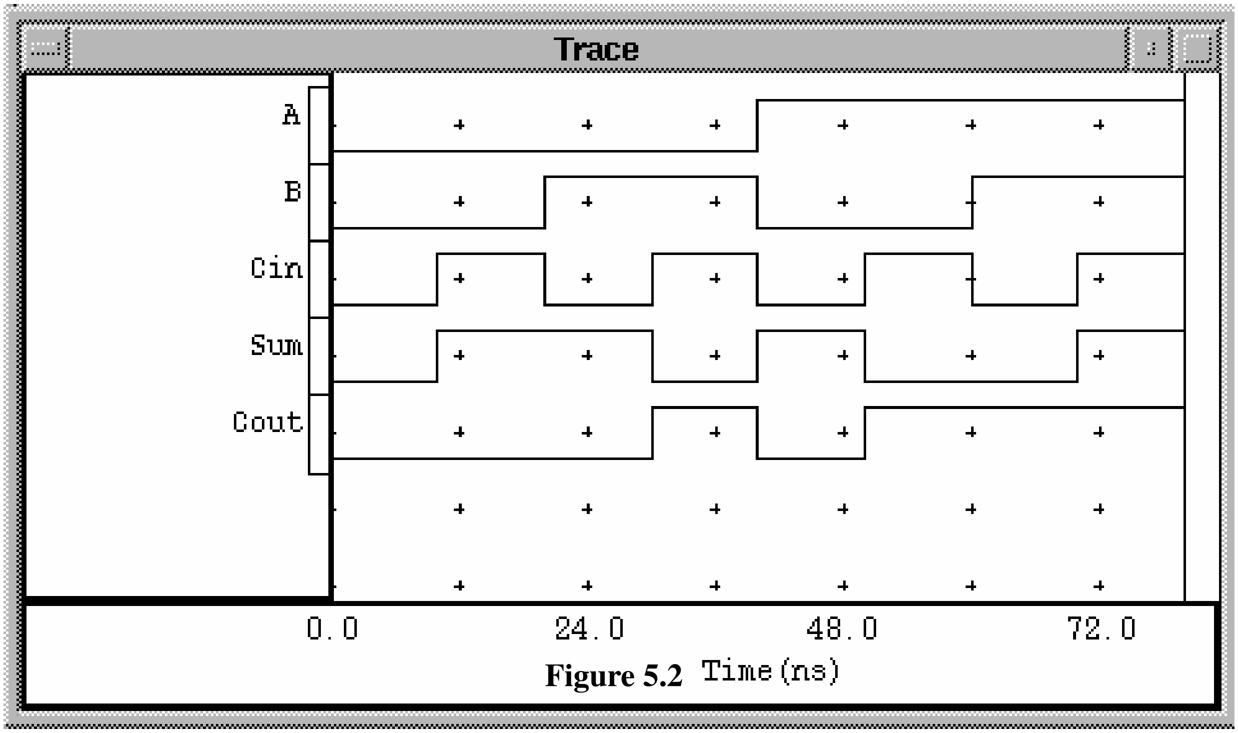


Figure 5.2