

# Introduction to Mentor Graphics

## Introduction

Mentor graphics is a tool that is used for digital design. It has many capabilities, varying from schematic capture to HDL (Hardware Description Language). It also allows simulating the circuit and verifying the design.

The goal of this tutorial is to provide first-time users with the basic capabilities needed to use the tool. For your first lab, you may follow the following steps:

- I. Log in to your account.
- II. Launch a command window and type da, which is going to run Design Architect.

## Design Architect

Design Architect is a schematic capturing tool for Mentor Graphics. In other words, it is a drawing program that knows the electrical components, wires, and electrical rules. The following figure shows the window of Design Architect.

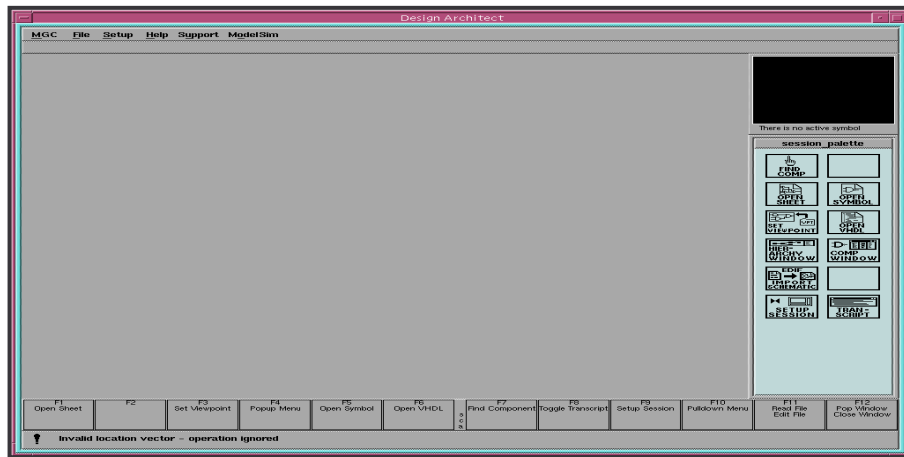
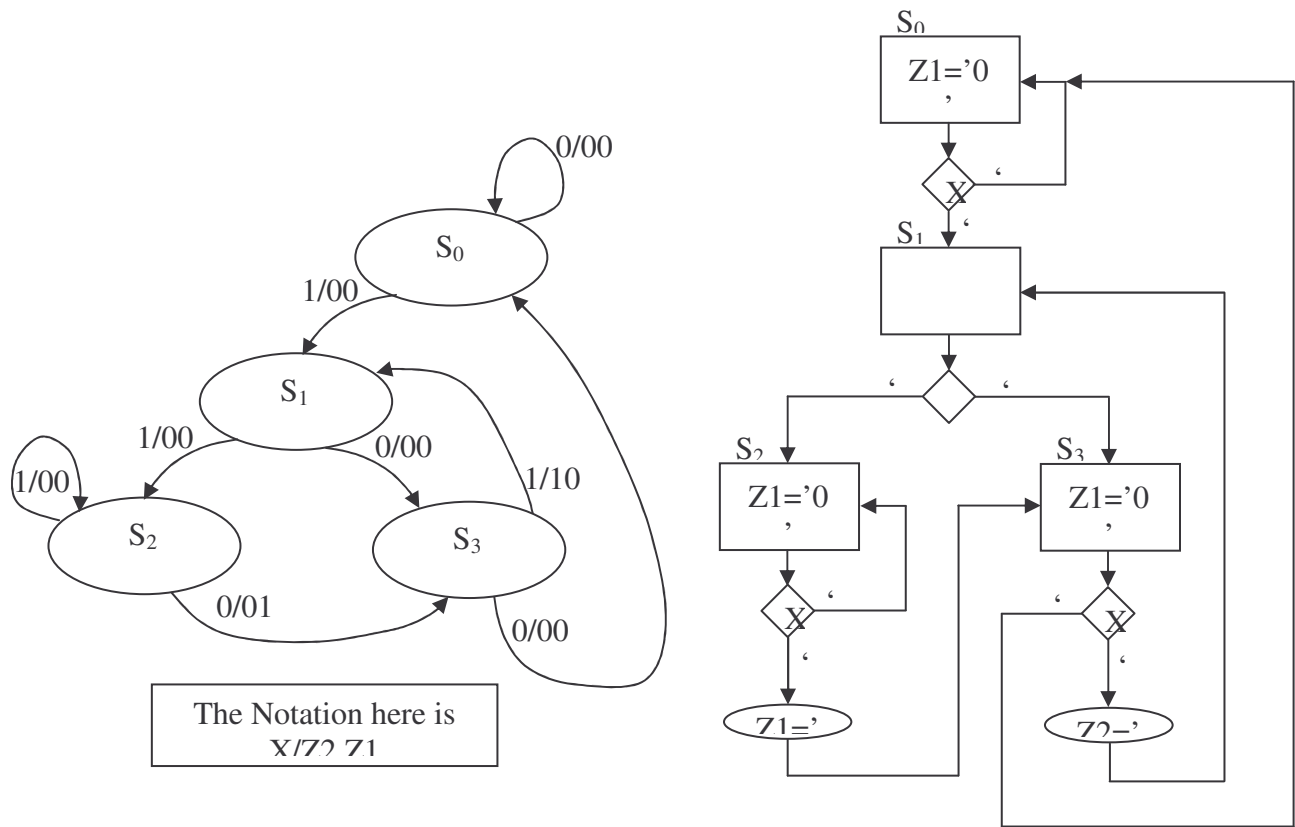


Figure 1

By maximizing the window, a pallet of icons becomes visible at the right hand side of the window. Pull down menu items are selected using the LMB (Left Mouse Button), and the pop-up menu is activated by clicking on the RMB (Right Mouse Button). Moreover, Mentor graphics has Softkeys, which allow users to perform operations using the keyboard.

## Schematic Editor

Basically, the schematic editor allows users to layout schematics of digital/analog circuits. In this lab, we consider an FSM that detects an input (X) and has two outputs (Z1) and (Z2). If the sequence “110” is detected then (Z1) is asserted. (Z2) is asserted if the sequence “101” is detected. Overlapping is to be taken into consideration. Two ways of representing this FSM are shown in Figure 2.



**Figure 2. Two ways of representing the FSM**

Table 1 represents the next state and outputs of the FSM, and Figure 3 represents the schematics of this FSM.

		X	
		0	1
Q <sub>0</sub> Q <sub>1</sub>			
S <sub>0</sub>	00	00,00	01,00
S <sub>1</sub>	01	10,00	11,00
S <sub>2</sub>	11	10,10	11,00
S <sub>3</sub>	10	00,00	01,01

**Table 1**

Following are the equations governing this FSM:

$$Q_0^* = Q_1 \quad \text{Equation 1}$$

$$Q_1^* = X \quad \text{Equation 2}$$

$$Z1 = Q_0 \cdot Q_1 \cdot \bar{X} \quad \text{Equation 3}$$

$$Z2 = Q_0 \cdot \bar{Q}_1 \cdot X \quad \text{Equation 4}$$

The asterisk appended to  $Q_0$  and  $Q_1$  indicates the next state.

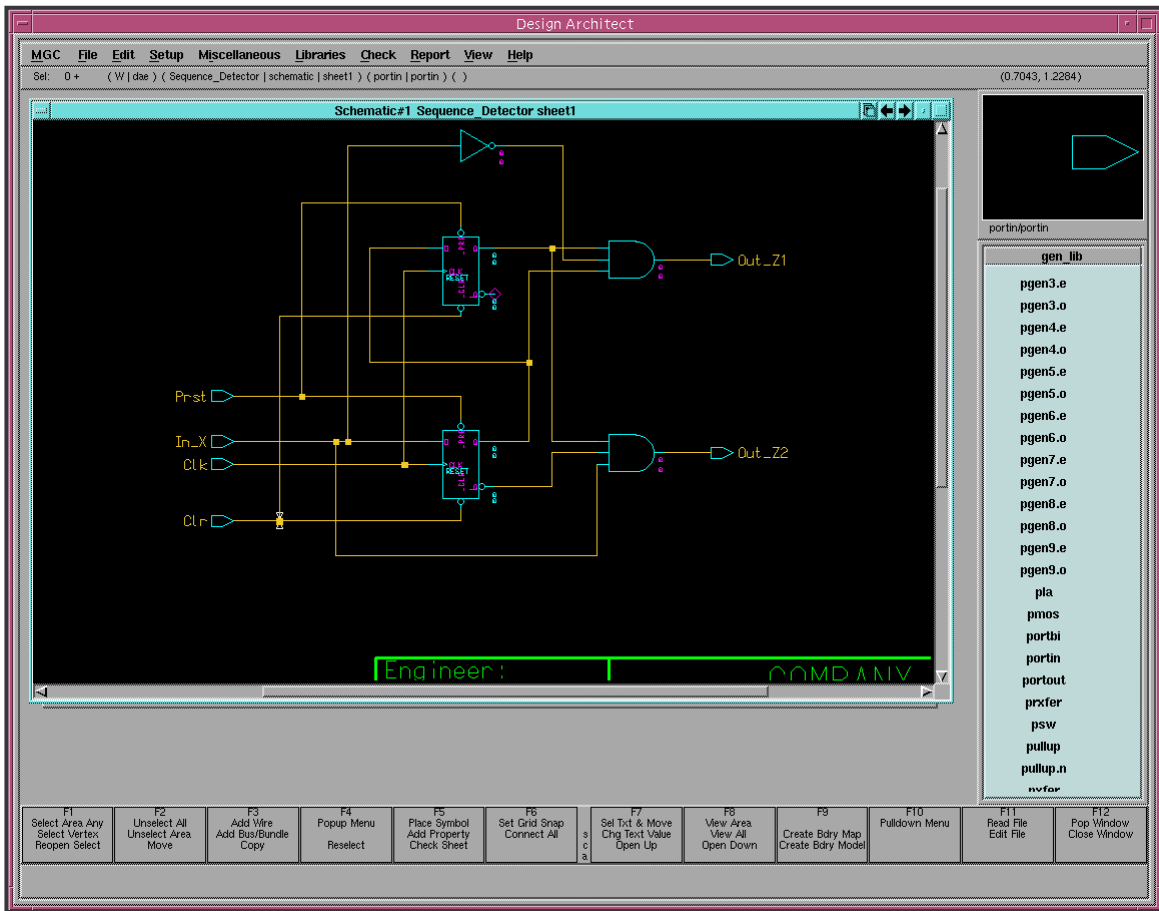


Figure 3

The following steps will guide you through this lab.

Click on the OPEN SHEET icon in the palette tray, or you may select File>Open>Sheet from the pull down menu bar, or alternatively press F1. A dialogue box is to appear. Append the name “/Sequence\_Detector” to the path shown in the Component Name field as shown in Figure 4.

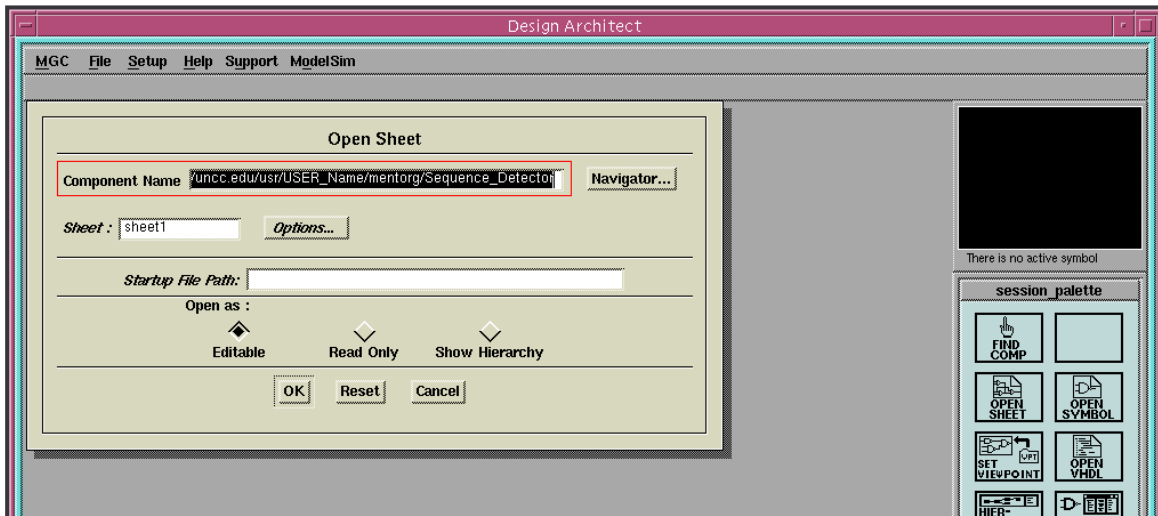


Figure 4

By clicking on the options button another dialog box opens. Select the New Sheet option.

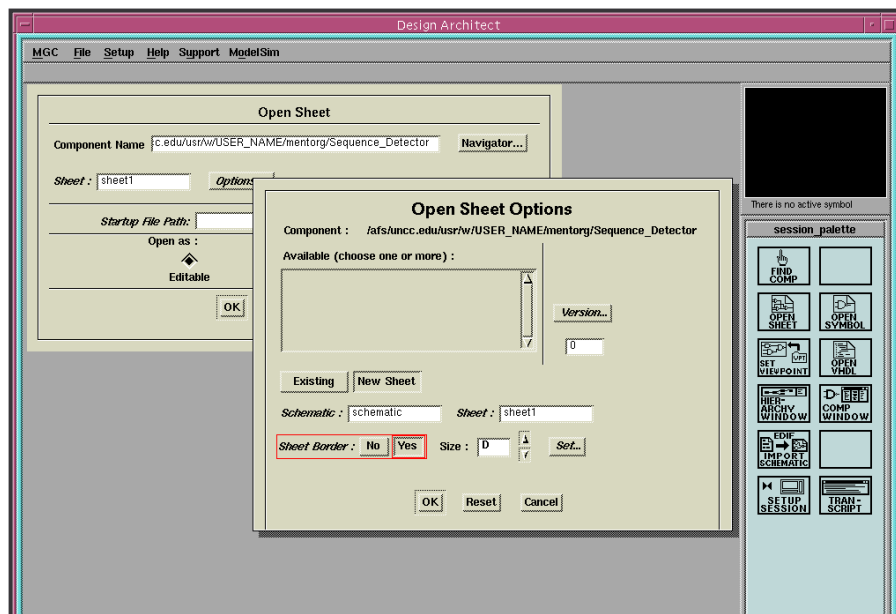


Figure 5

Click the OK button for the Sheet Border option as shown in Figure 5, and set sheet size to B (or you may press the Set button, see Figure 6).

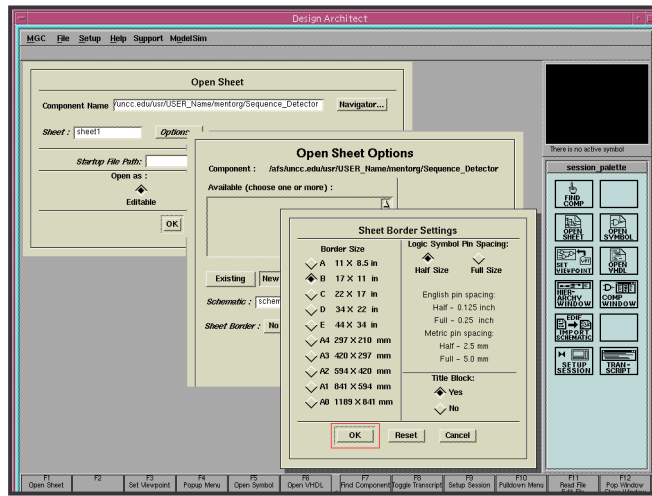


Figure 6

Another dialog box titled “Title Block Information” will show up. You may set the Revision Level to A, and the Drawing Number to 1. To start drawing your schematic, click on Libraries>MGC Digital Libraries>Display Libraries Palette, then select “gen\_lib”. By doing so, the gates and circuit components available in this library will be visible in the right hand side palette (See Figure 7).

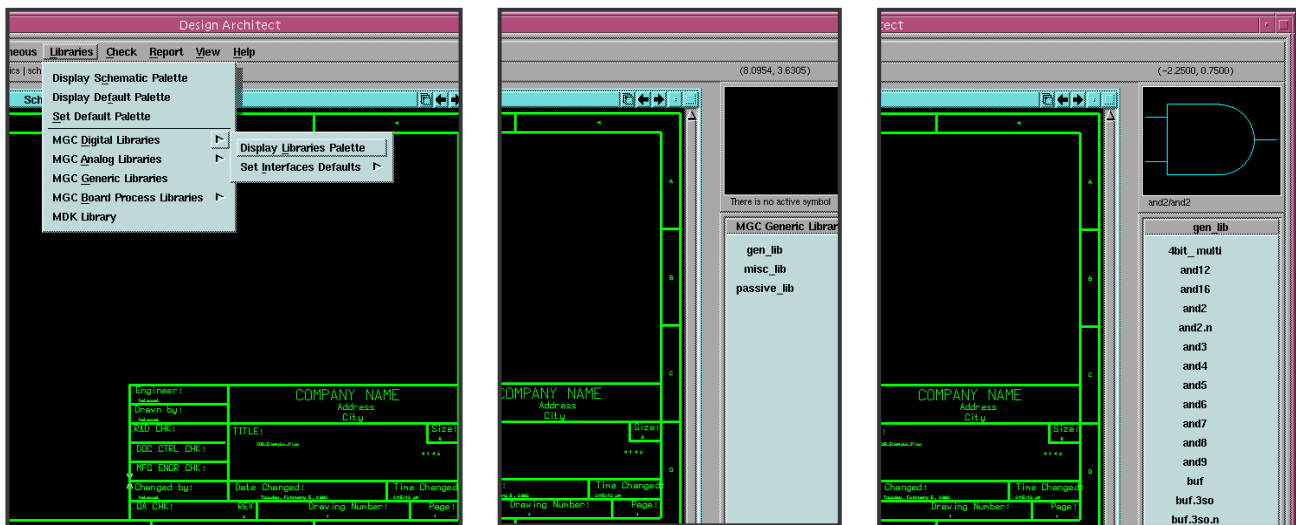


Figure 7

Now you can scroll through the listed components. Once you find the component you want to add to your schematic, click on it once and its symbol should show up in the

small window at the upper right hand side of your screen as shown in Figure 8. By moving the cursor into the drawing area, the cursor changes to look the same as the desired symbol, **click only once** where you want to add the symbol, otherwise you will have two components overlapping. Hence, the error check step that you will perform by the end of the circuit layout will fail. Repeat this step for all the required components in the schematic (See Figure 8). The FF used in this lab is a D-ff with reset.

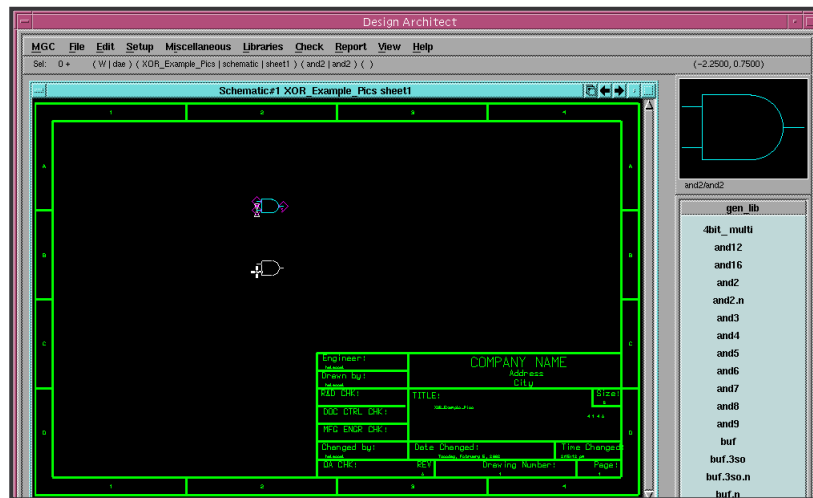


Figure 8

To connect these components together with wires, you need to press F3. After that you click on the first node to connect and drag the line towards the destination node, then double click on the destination node to stop wiring. If you need to bend the wire, then **click once** and change your routing direction.

The next step of drawing the schematic is to add ports to the circuit. Find the “portin” and “portout” components from within the components palette. Add these elements to your schematic the same way you add normal components and connect them to the circuit ports by wires.

The final step in building the circuit schematic is to name the ports. You can do that by clicking once on the desired “NET” on the schematic, then press “Shift+F7”. This will

launch a window; you need to print the name of the port in the “New Value” field. For our first lab, you may use Clk, Prst, Clr, In\_X, Out\_Z1, and Out\_Z2.

Before being able to simulate your circuit, you need to make sure that it is free of errors. This can be done by using the pull down menu and selecting Check>Sheet as shown in Figure 9.

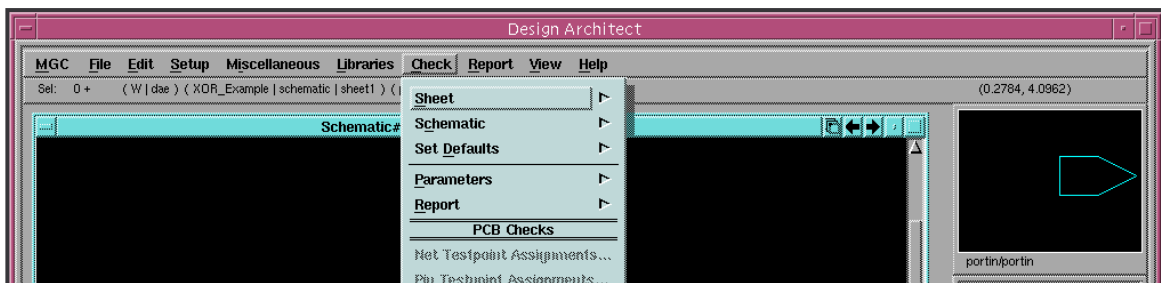


Figure 9

If the design is error-free then the check result window should look like the one shown in Figure 10.

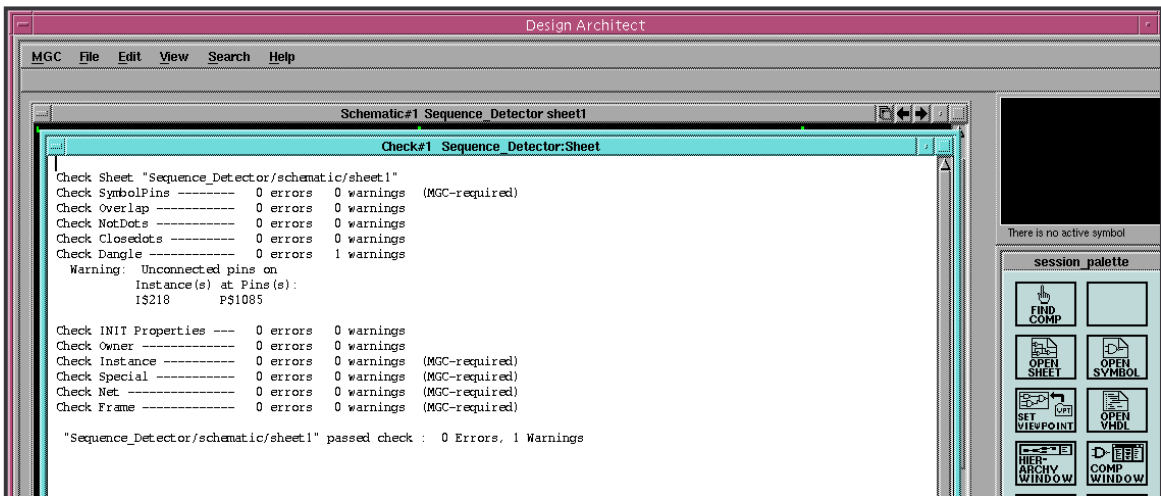


Figure 10

Before quitting the Design Architect, you need to save your design by choosing File>Save Sheet from the pull down menu.

## Simulating the circuit

The basic idea here is to apply input stimuli to the circuit such that its functionality can be verified by observing its output response. First you need to change the current directory to the one containing your design (type `cd mentorg` in the command line), and then you can launch the simulator by typing “quicksim” from within the command window. After that you can open the design to be simulated using the pull down menu `File>Open Sheet`, as shown in Figure 11.

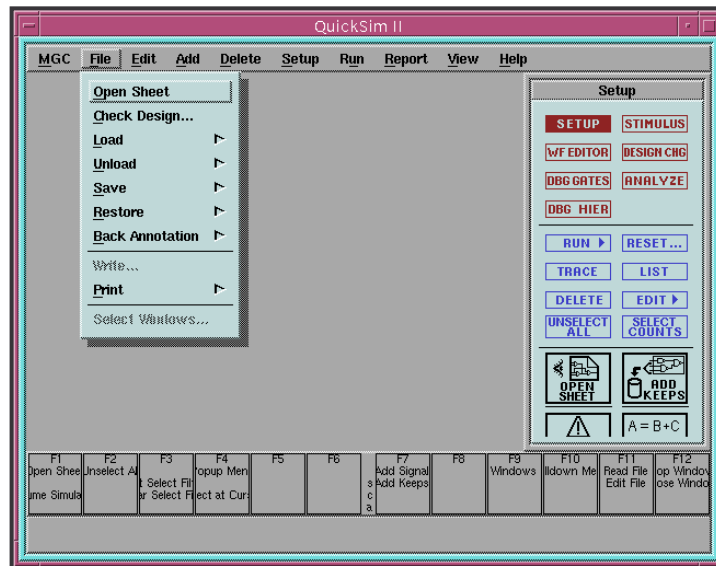


Figure 11

By selecting your circuit for simulation (for this lab it is `Sequence_Detector`), the window should look like that of Figure 12.

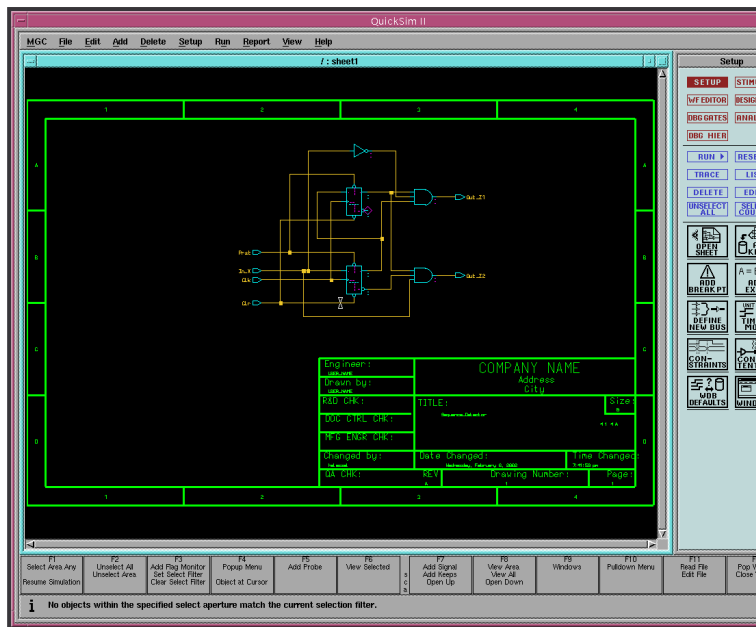


Figure 12

Now you need to force the inputs of your circuits to the logical values of '0' and '1' over a period of time and observe the output. The input stimuli need to be developed such that they cover all possible states and state transitions (verify every branch in the FSM) in order to make sure that the circuit is functioning correctly.

First you need to click on the STIMULUS option in the setup palette that is shown in Figure 12. The palette will change the items shown in it, and now you can choose the option Add Force, and a window that is similar to the ones shown in Figure 13 will be invoked.

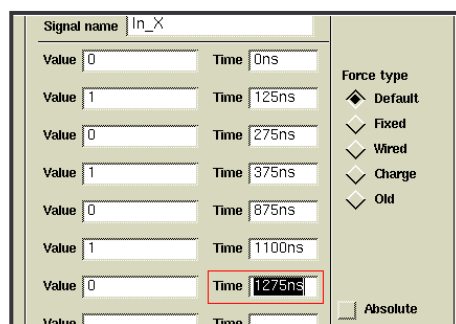
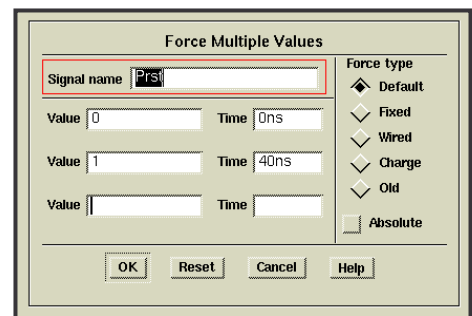
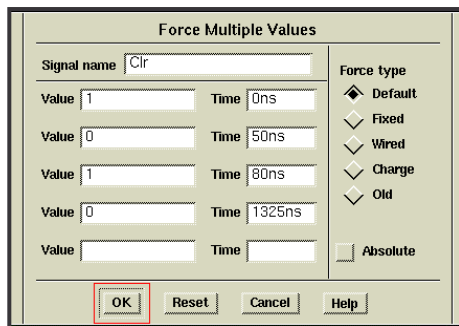
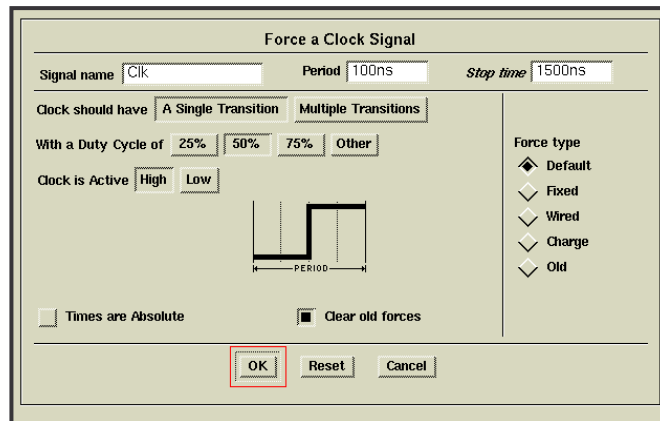


Figure 13

Notice the way the stimulus vectors for the inputs In\_X and Clr are written. For the input In\_X for example, the input stimulus means, apply '0' at 0ns, '1' at 125ns, '0' at 275ns, '1' at 375ns, '0' at 875ns, '1' at 1100ns, and '0' at 1275ns. After that you need to select the ports to be observed when the input stimuli are to be applied. To do so, click on the

option TRACE in the palette tray, and enter the names of ports to be observed as shown in Figure 14. Alternatively, you may highlight the ports to be observed using the LMB.

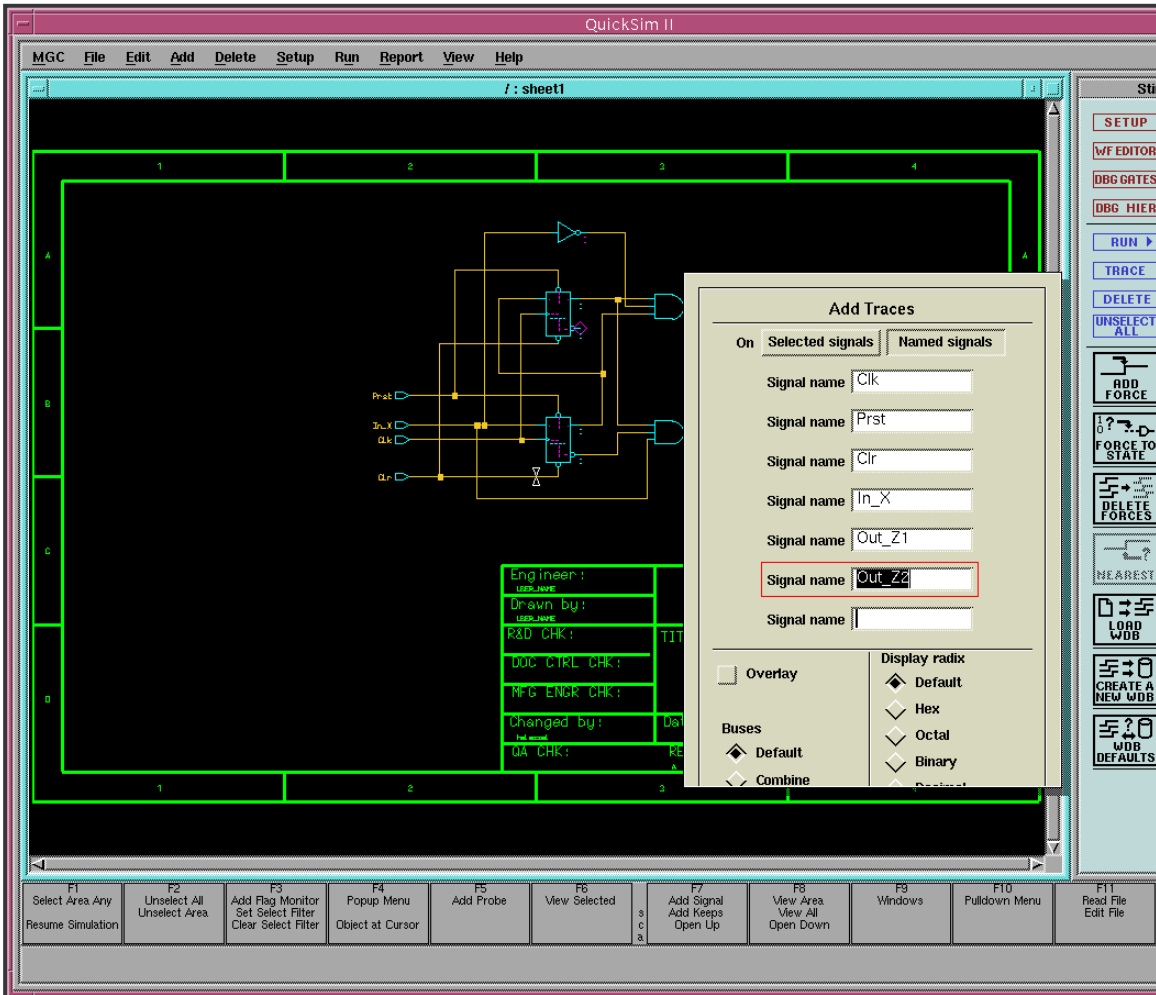


Figure 14

The next step is to define the time over which the input stimuli will be applied. This can be done by selecting the RUN option in the palette tray and entering the test time. You need to type the desirable test time, which is at least equal to the period of time you chose for the inputs' stimulus, without appending (ns) to it. After that you can launch the test and observe the output by choosing the RUN option. The result should look similar to the one shown in Figures 15 and 16. Notice that you may develop your own test stimulus values.

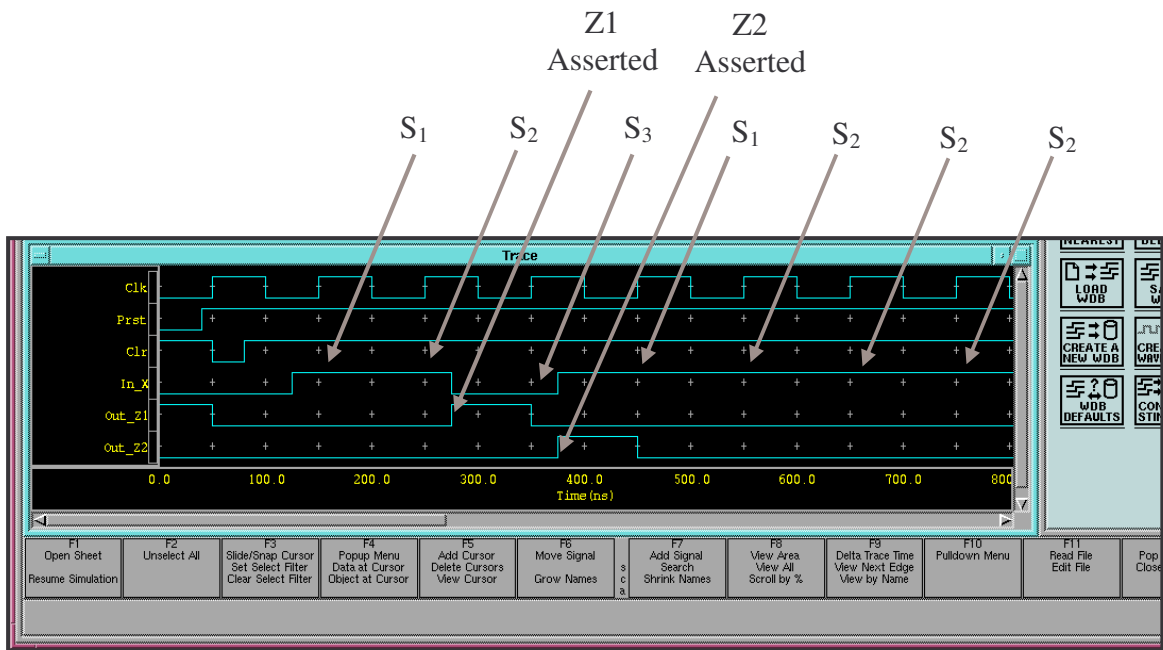


Figure 15

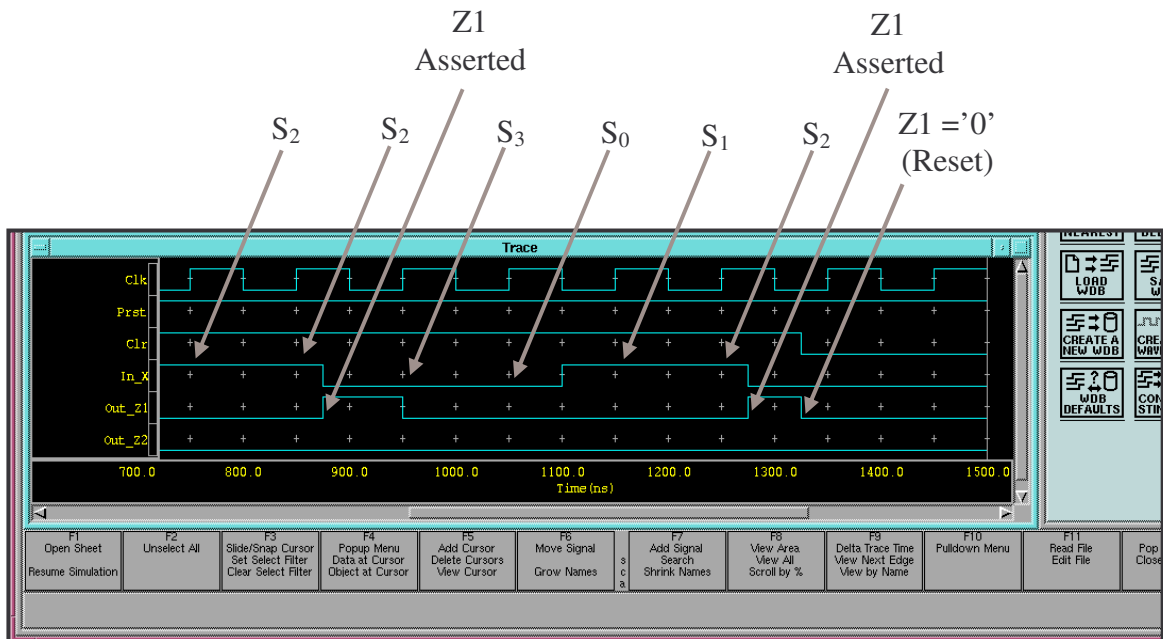


Figure 16

If you want to include a graphical representation of your circuit schematic and the simulation results, you can use the XV software, to do that type xv in the command window. To exit the Quick Sim press MGC>Exit, and select the “Without Saving” option.