Lab 3: Multisim & Ultiboard

INTRODUCTION

Multisim is a schematic capture and simulation application that assists you in carrying out the major steps in the circuit design flow. Multisim can be used for both analog and digital circuits and also includes mixed analog/digital simulation capability, and microcontroller co-simulation. Simulating the circuits before building them, catches errors early in the design flow, saving time and money. The Multisim's user interface and its main elements can be seen in Figure 1.
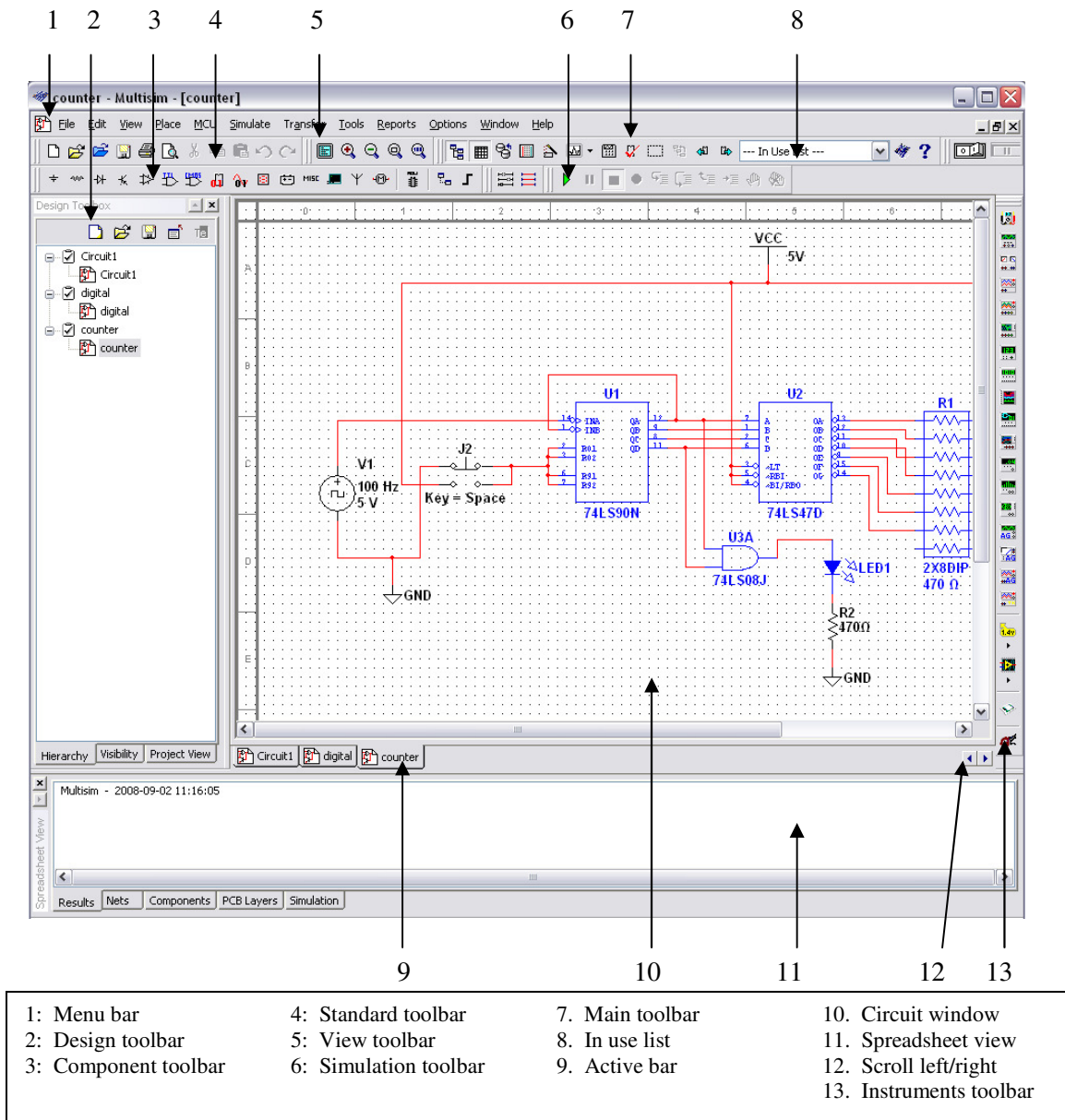

1: Menu bar	4: Standard toolbar	7. Main toolbar	10. Circuit window
2: Design toolbar	5: View toolbar	8. In use list	11. Spreadsheet view
3: Component toolbar	6: Simulation toolbar	9. Active bar	12. Scroll left/right
			13. Instruments toolbar

Figure 1. Multisim's interface

Ultiboard, fed from Multisim, is used to design printed circuit boards, perform certain basic mechanical CAD operations, and prepare them for manufacturing. Ultiboard also provides automated parts placement and layout.

LAB Exercise 1 – Digital Circuit

Objective:
Place, wire and simulate a digital circuit.

Procedure

In this exercise, you will place and wire the components in the circuit shown below.
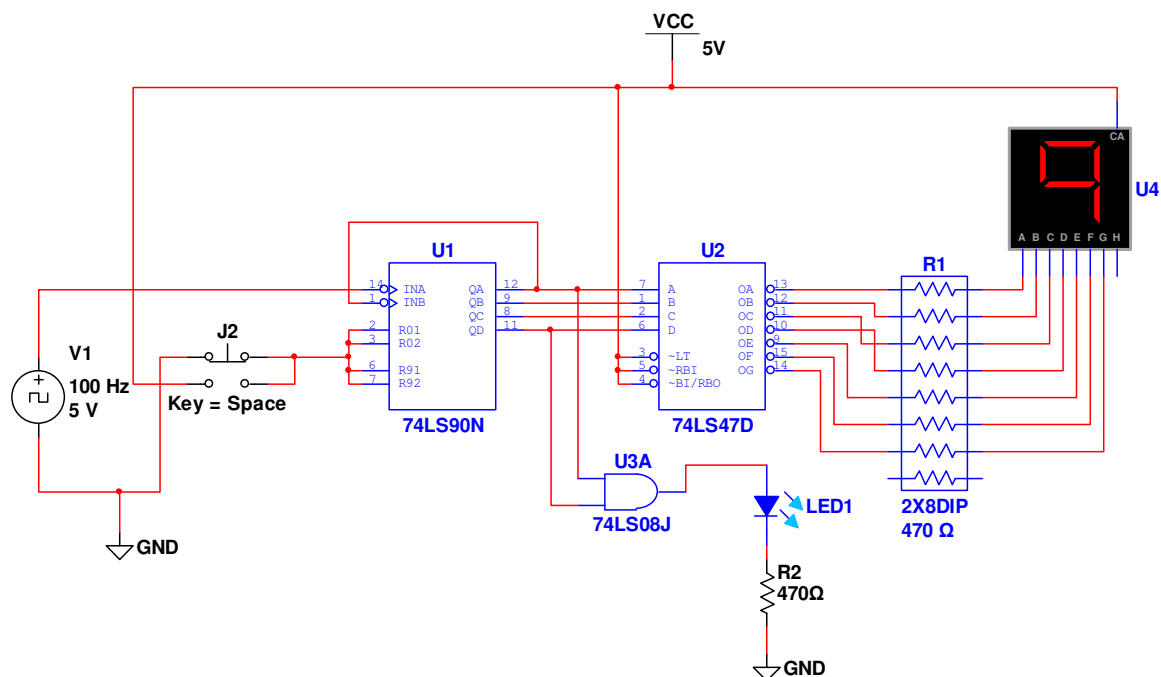

Figure 2. BCD counter

1. Select **Start » All Programs » Applications » National Instruments » Circuit Design Suite 0.1 » Multisim 10.1**. A blank file opens on the workspace called Circuit1.

2. Select **File » Save As** to display a standard Windows Save dialog. Name and save the file in your Z drive.

3. Select **Place » Component** to display the **Select a Component** browser. Select the group **Indicators** in the **Group** menu, and the **Hex_display Family**. Navigate to the 7-segment LED display as shown below and click OK. The component appears as a "ghost" on the cursor.
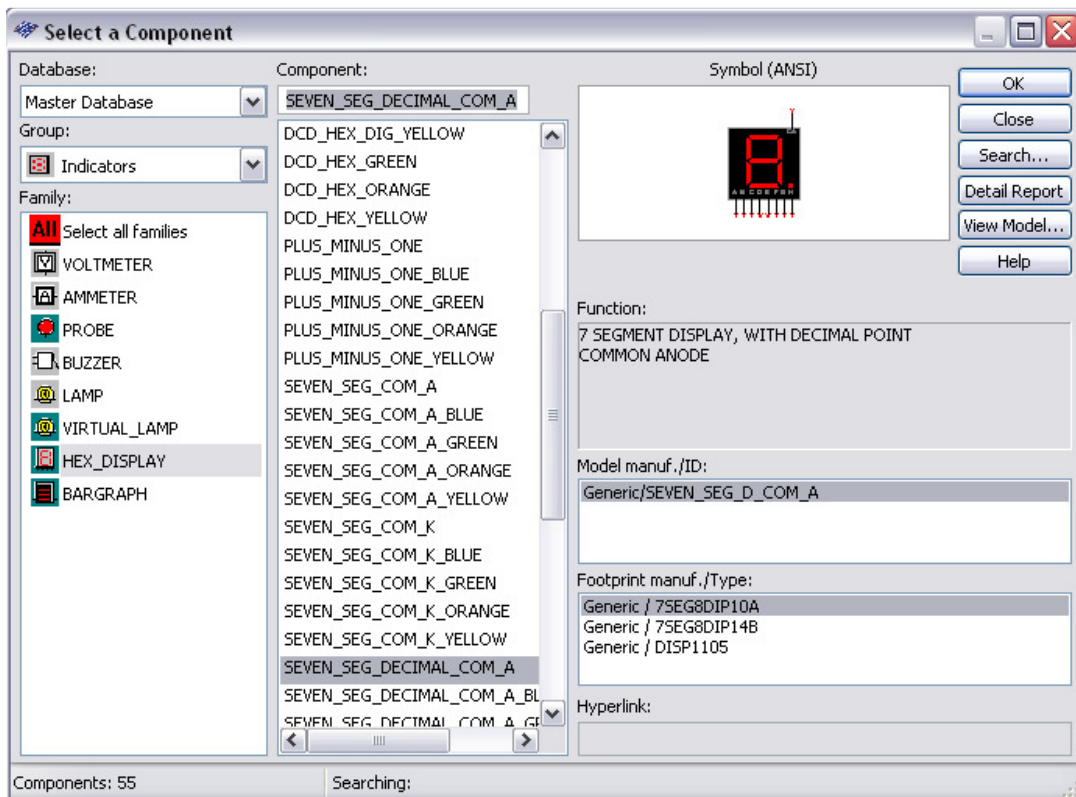
Figure 3. Select a Component window

4. Place the remaining components in the workspace. The LED is in the **Diode Group**. The voltage sources are in the **Sources Group**; Power sources family and Signal voltage sources family. The resistors are in the **Basic Group.** The integrated circuits are in the **TTL Group**. The pushbutton is in the **Electromechanical Group**; Momentary switches family.

   Note: as this is a digital circuit, you should use DGND as ground. For analog circuits use the GROUND component.

5. You can adjust the properties of the components, such as the frequency of the square wave, double clicking on them. Right click on a component to modify its orientation.

6. To wire the circuit, click on a pin on a component to start the connection (your pointer turns into a crosshair) and move the mouse. A wire appears, attached to your cursor. Click on a pin on the second component to finish the connection. Multisim automatically places the wire, which conveniently snaps to an appropriate configuration. You can also control the flow of the wire by clicking on points as you move the mouse. Each click "fixes" the wire to that point.

7. Select **Simulate»Run** or press the Run button ▶ in the simulation toolbar to simulate the circuit. As the circuit simulates the 7-segment display counts up and the LED flashes at the end of each count cycle. The pushbutton is an interactive component. Look what happen when it is pressed.

8. Save and close the circuit.

END OF EXERCISE 1.

LAB Exercise 2 – Transfer from Schematic to PCB

OBJECTIVE:
Transfer the schematic designed in Exercise 1 to a useable PCB layout using Ultiboard.

Procedure

1. Open the saved file from the end of Exercise 1.

2. Transfer the schematic design to Ultiboard 10 by selecting **Transfer » Transfer to Ultiboard 10**.

Figure 4. Transfer Menu

3. Use the **Save As** dialog box to save the new Ultiboard file to your **Z** drive. Click **OK** on the message box that appears.

4. Accept the default trace width and clearance that is given when Ultiboard opens. Make sure that all fields are selected in the Import Netlist Action Selection window and click **OK**. The program should look like Figure 5.
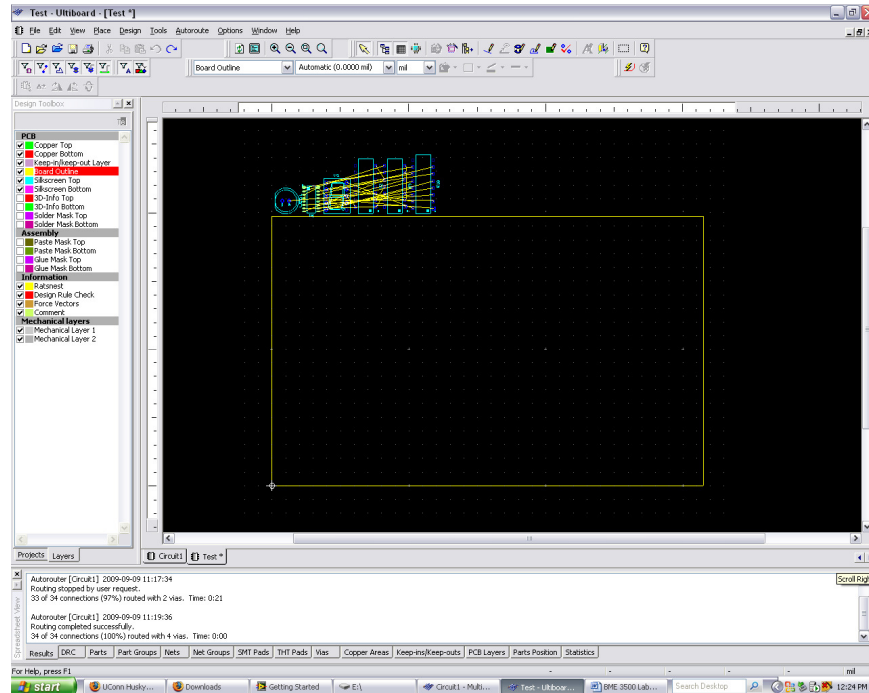

Figure 5. Voltage limiter

5. Select the **Layers** tab near the middle of the left of the screen. This lists the board layers and other useful elements of the board. Double-clicking a particular layer sets that layer as active. Double-click **Board Outline** from the list.

6. Right-click an edge of the yellow rectangle and select **Properties** from the menu that appears. Under the **Rectangle** tab set the width to 2000 and the height to 1500. Click **OK** and notice that the yellow rectangle becomes much smaller. This rectangle represents the area of the circuit board in which all of the components and traces must fit.

7. The next step to PCB design is to layout the components. Double-click the checkbox next to **Ratsnest** in the layers list. This removes the yellow lines that connect all of the components together and simplifies the picture.

8. Begin placing the components inside of the yellow rectangle. Notice single brown lines coming out of each component. These are **Force Vectors** and they point to the most efficient place to put the component based on its necessary connections. Use these as a guide for placement, but be careful because **Force Vectors** would

ideally have all of the components stacked on top of each other in the middle of the board.

9. There are many different ways this board could be laid out and it takes practice to make efficient boards. Use Figure 6 as an example of where to put components, or feel free to create your own unique layout.
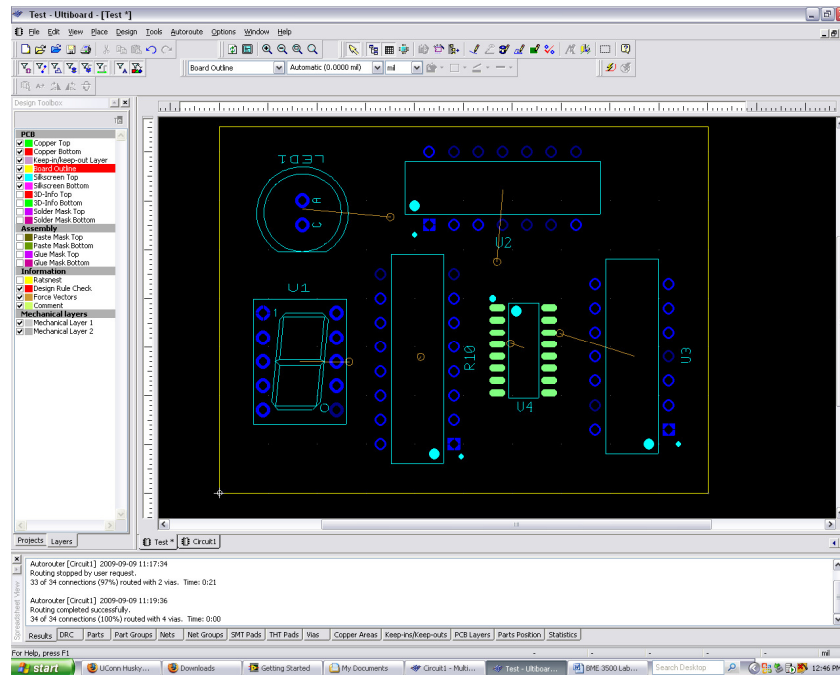

Figure 6. Possible Component Layout

10. The next step is setting the up the routing parameters for laying out the copper trace. Go to the **Netlist Editor** by selecting **Tools » Netlist Editor**. This tool keeps track of all connections between pins on the board. Each set of pins connected together are referred to as a Net. Use the dropdown menu in the **Netlist Editor** to view the different Nets on the board. The tabs in the **Netlist Editor** provide different information and options for the selected net.

11. To prepare the Nets for routing select the first Net from the list. Select the **Misc** tab and check the box next to **Copper Bottom**. This tells the Autorouter that this Net can be connected by placing trace either on the top of the board or the bottom of the board.

12. In this case it is best if the Autorouter can use the top or bottom layers for all of the Nets. To save time select the **Nets** tab from the group of tabs at the bottom of the screen. Right-click one of the rows and choose **Select All** from the menu that appears. Click inside one of the cells in the **Routing Layers** column. Check the box next to **Copper Bottom** in the window that appears and click **OK**.

13. The board layout is ready for the Autorouter. Start the Autorouter by selecting **Autoroute » Start/Resume Autorouting**. The Autorouter should only take

seconds to place traces for the board. Notice that there are green and red lines that appear. The green lines represent copper trace on the top layer of the board and red lines represent copper trace on the bottom layer of the board. Figure 6 shows an example of what the board from Figure 5 looks like after routing.
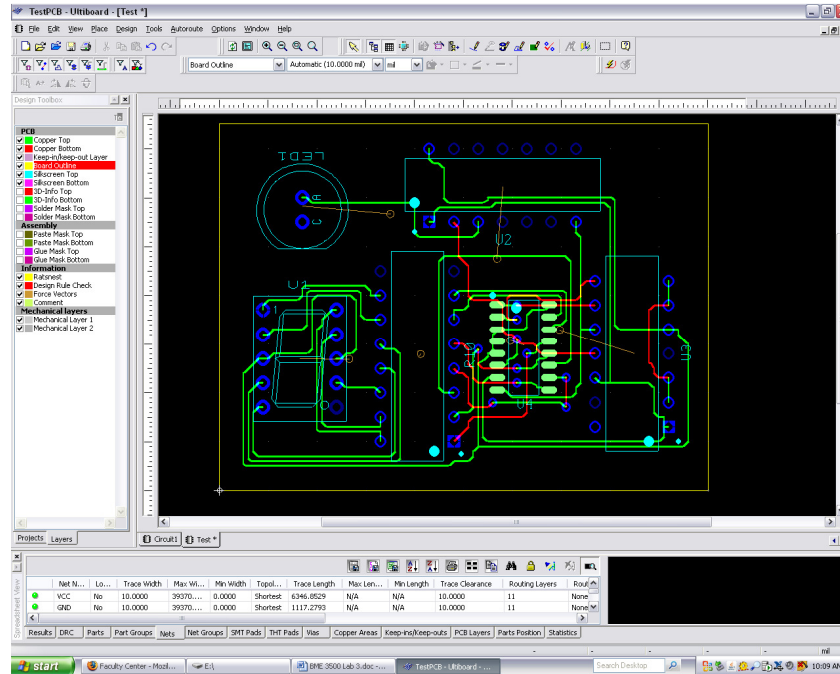

Figure 6. Routed PCB

14. It is important to cleanup and check the work of the Autorouter after use. Excessive bends and sharp angles in the traces greatly increase the chances of board failure during manufacturing and use. Take some time to look through the traces and get a feel for dragging traces into more appropriate shapes and angles. The Autorouter is never perfect and there are always traces that can be cleaned up.

15. Once you are satisfied with the traces on the board Save and Close the design.

END OF EXERCISE 2.