

Introduction to Multisim and Ultiboard

Contents

Multisim & Ultiboard.....	2
Introduction	2
Simulation in Multisim	3
Objective:	3
Procedure	3
Transfer from Schematic to PCB	10
Objective:	10
Procedure	10

Table of Figures

Figure 1: Multisim's user interface	2
Figure 2: Half-adder schematic diagram	3
Figure 3: Launch Multisim / Ultiboard.....	3
Figure 4: Placing components, 2-INPUT XOR	4
Figure 5: Placing component, RED LED.....	5
Figure 6: Placing component, VCC (TTL Supply)	6
Figure 7: Placing component, resistor	7
Figure 8: Placing component, dip switch	8
Figure 9: Run simulation	9
Figure 10: Simulation of the half-adder with A=1, B=0 where Sum=1, Carry=0 (LED1 in red means turn-on)	9
Figure 11: Transfer to Ultiboard.....	10
Figure 12: Ultiboard interface.....	11
Figure 13: Footprint for Resistor	11
Figure 14: Board outline size	12
Figure 15: Placement of the components inside board area	13
Figure 16: Netlist editor, selecting Routing layers	14
Figure 17: Select all.....	15
Figure 18: Select Routing layers	15
Figure 19: PCB after auto-routing	17

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 will point out errors early in the design flow, save both time and money. The **Multisim**'s user interface and its main elements can be seen in Figure 1. **Ultiboard**, fed from **Multisim**, is used to design printed circuit boards (PCB), perform certain basic mechanical CAD operations, and prepare them for manufacturing. **Ultiboard** also provides automated parts placement and layout.

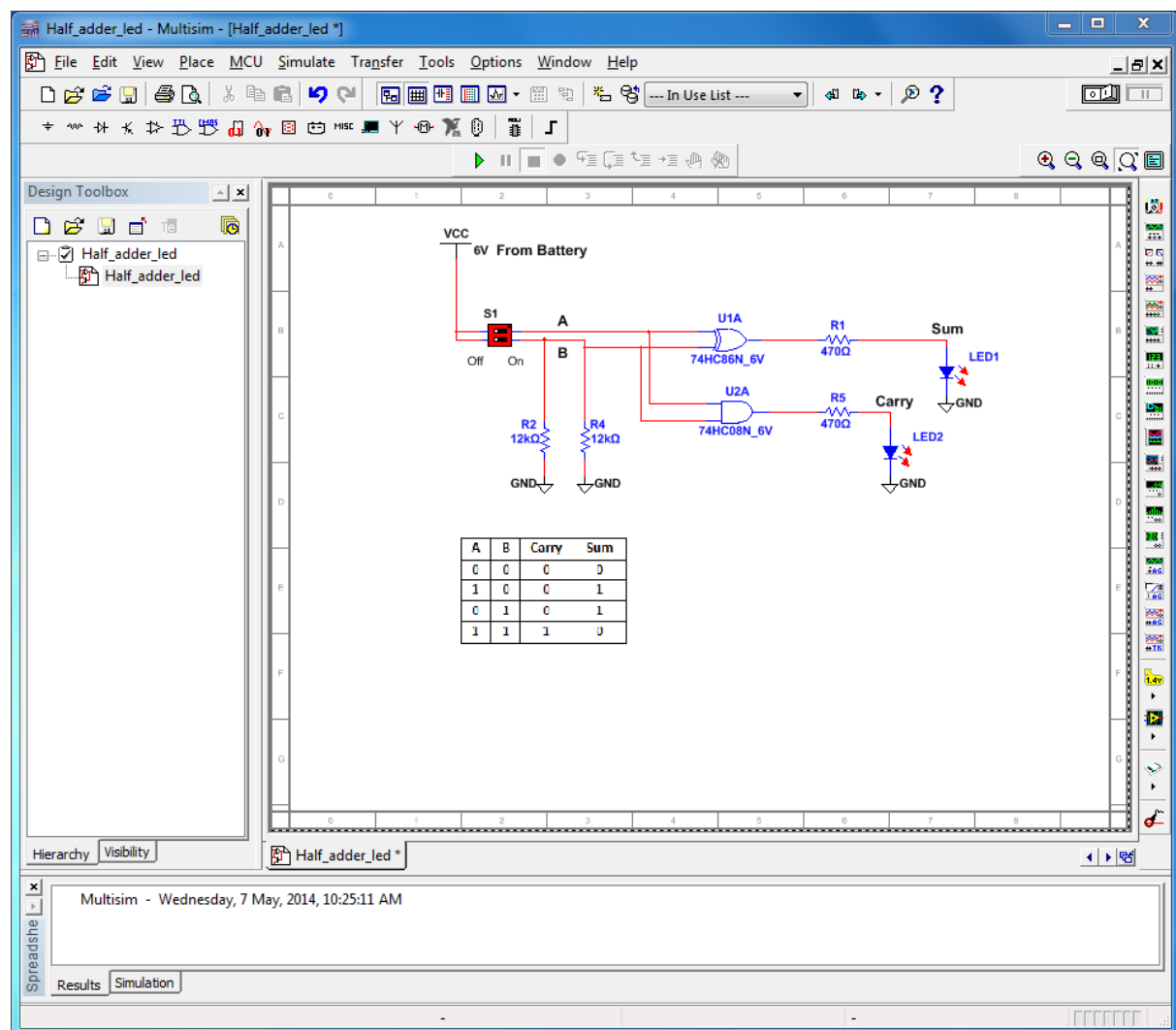


Figure 1: Multisim's user interface

Simulation in Multisim

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.

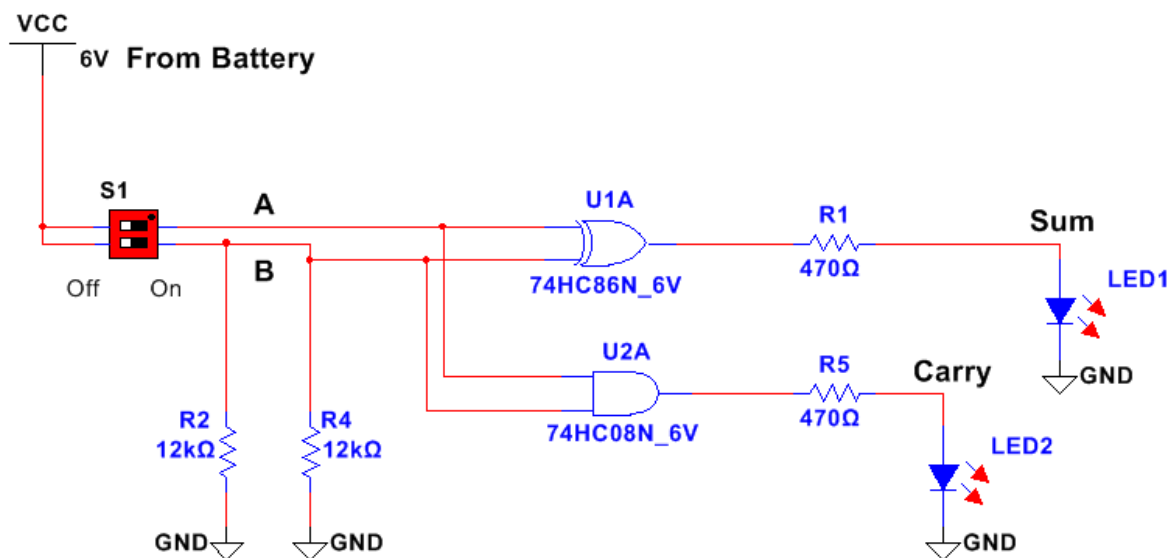


Figure 2: Half-adder schematic diagram

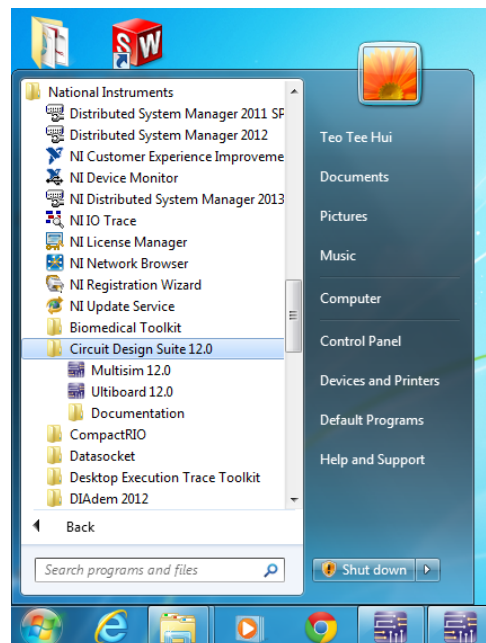


Figure 3: Launch Multisim / Ultiboard

1. Select **Start » All Programs » Applications » National Instruments » Circuit Design Suite 12.0 » Multisim 12.0**. A blank file opens on the workspace, just like Figure 1.
2. Select **File » Save As** to display a standard Windows Save dialog. Name and save the file in your preferred drive.
3. Select **Place » Component** to display the **Select a Component** browser. E.g. select the group **CMOS** in the **Group** menu, and the **74HC_6V Family**. Navigate to the 74HC86N_6V as shown below and click OK. The component appears as a “ghost” on the cursor.

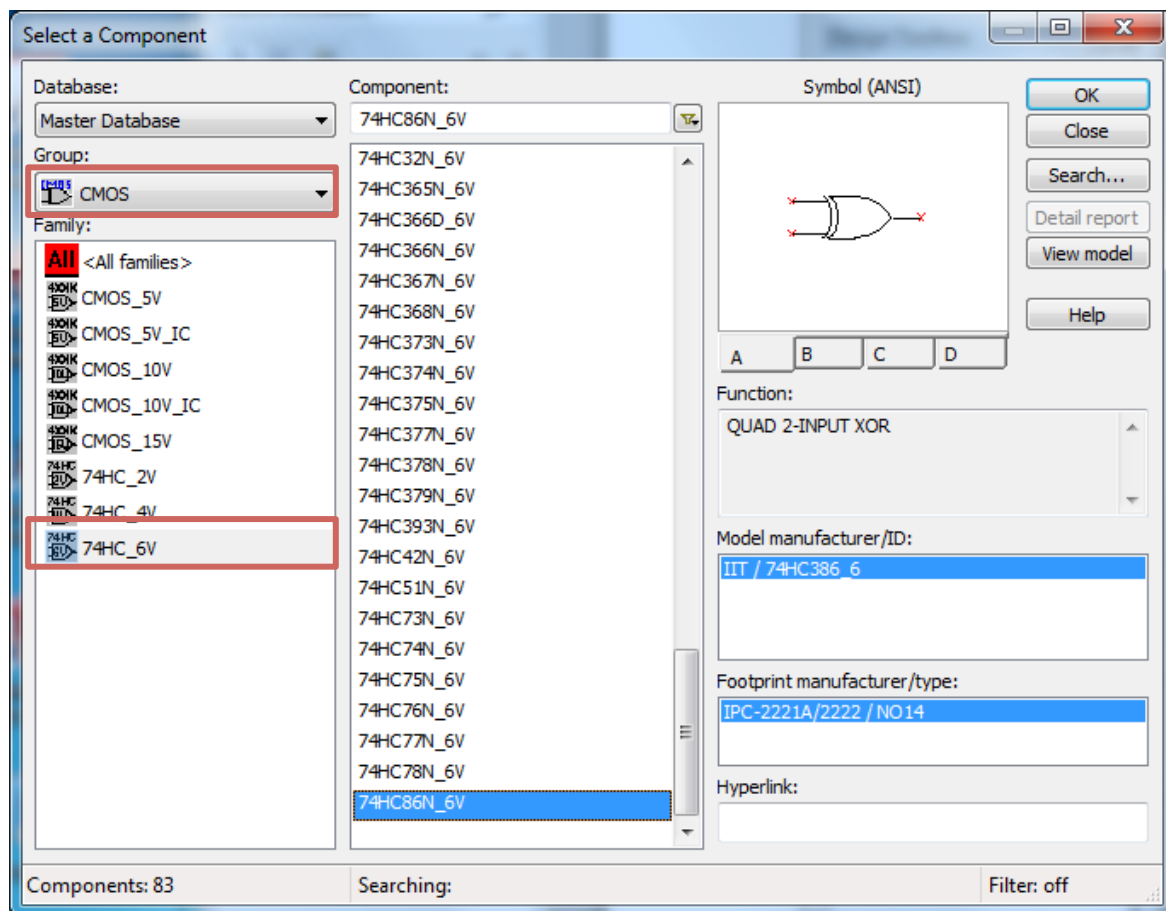


Figure 4: Placing components, 2-INPUT XOR

4. Place the remaining components in the workspace. The LED is in the **Diodes** 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 **CMOS** group. The switch is also in the **Basic** group (switches family).

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

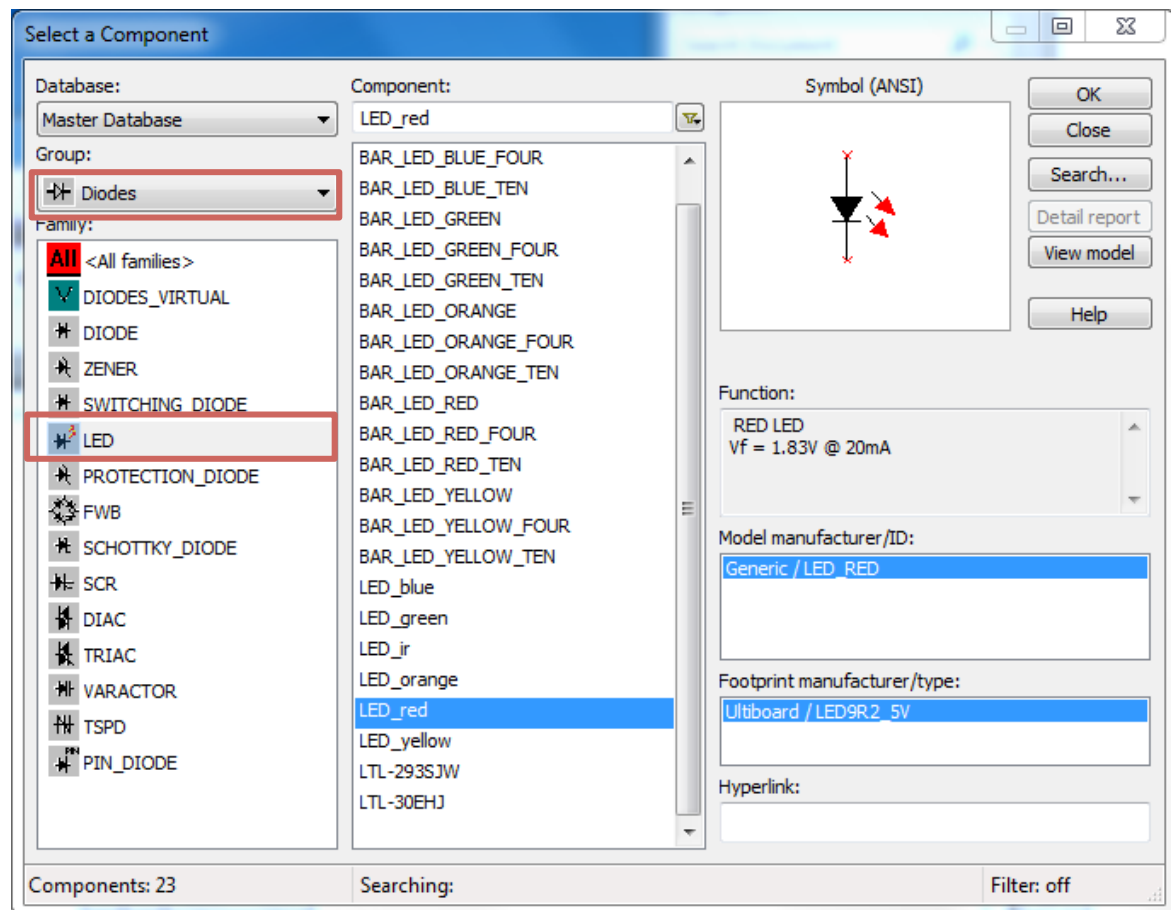


Figure 5: Placing component, RED LED

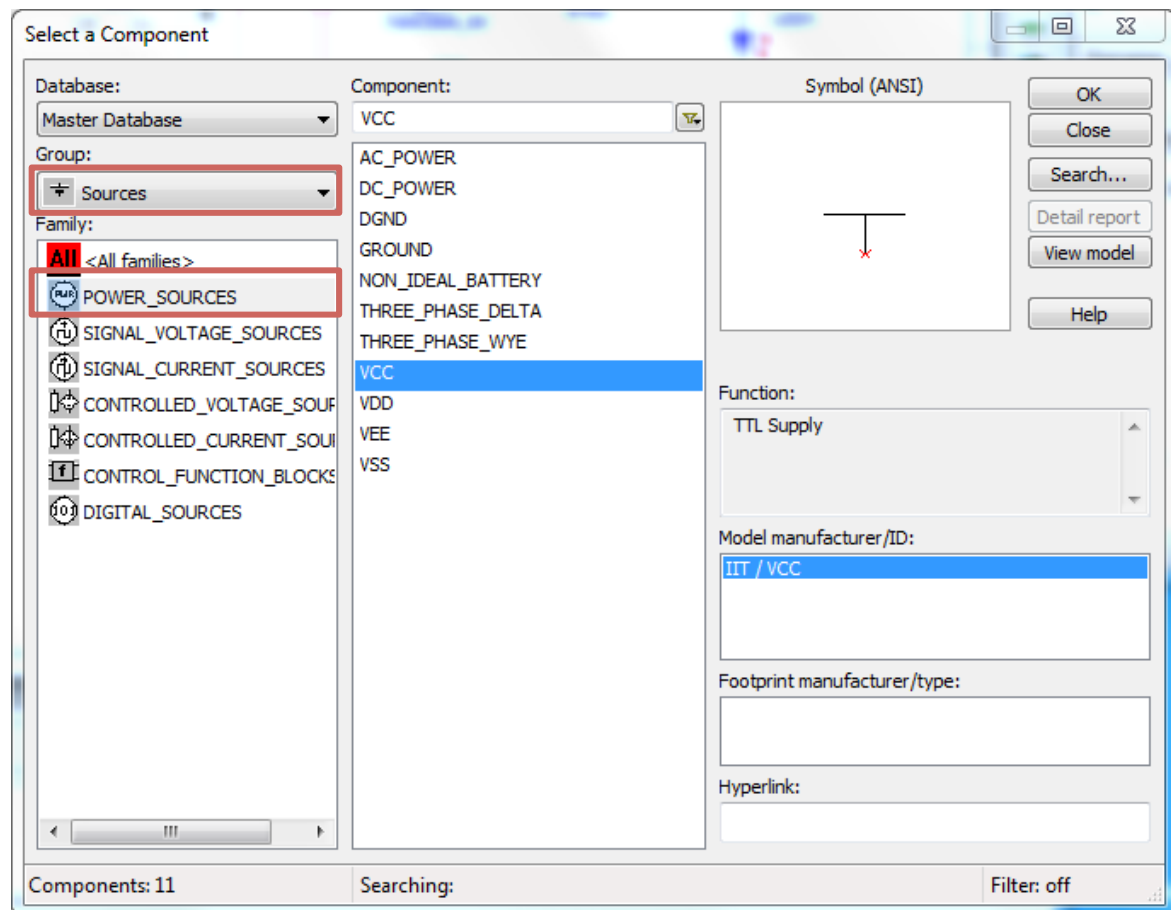


Figure 6: Placing component, VCC (TTL Supply)

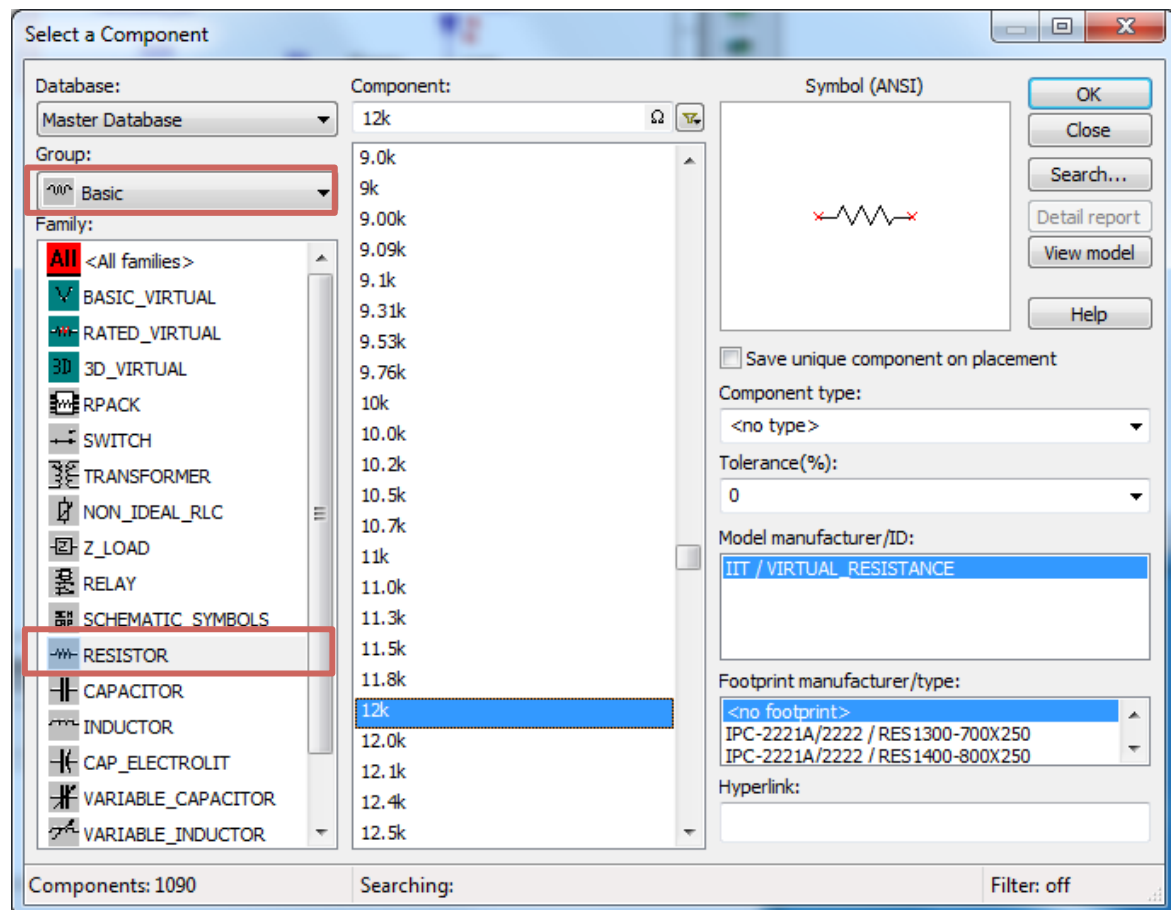


Figure 7: Placing component, resistor

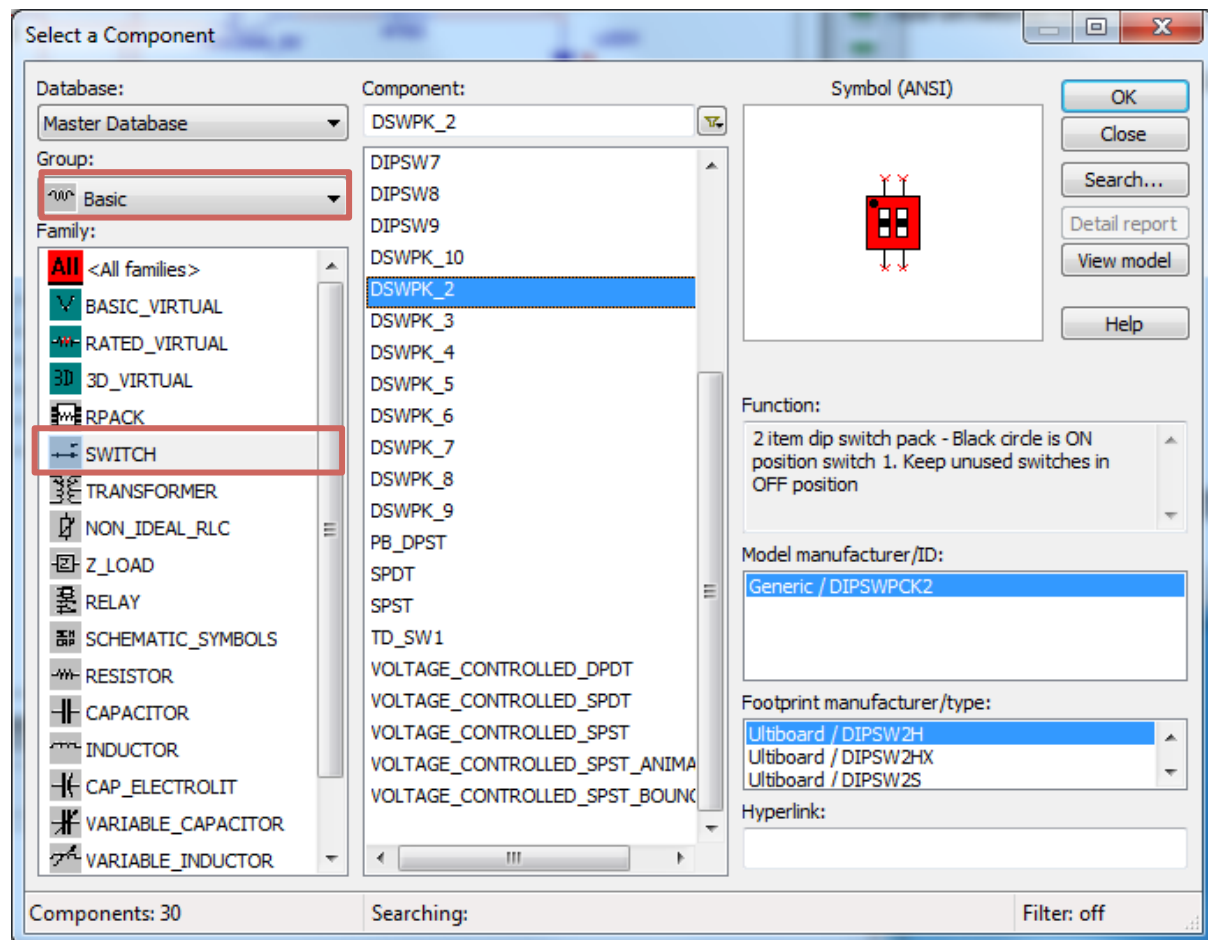
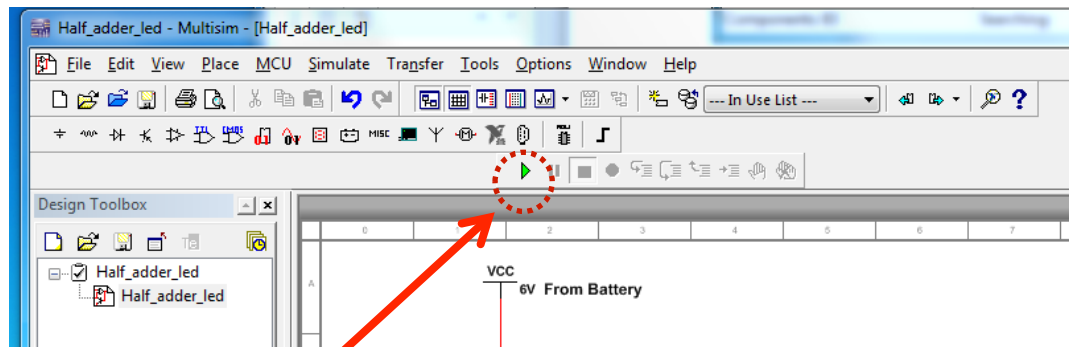


Figure 8: Placing component, dip switch

5. You can adjust the properties of the components by 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 half-adder, the LED lights up. The dip switch is an interactive component. Look at what happens when it is turned on/ off, Figure 10.



Run Simulation

Figure 9: Run simulation

Stop Simulation

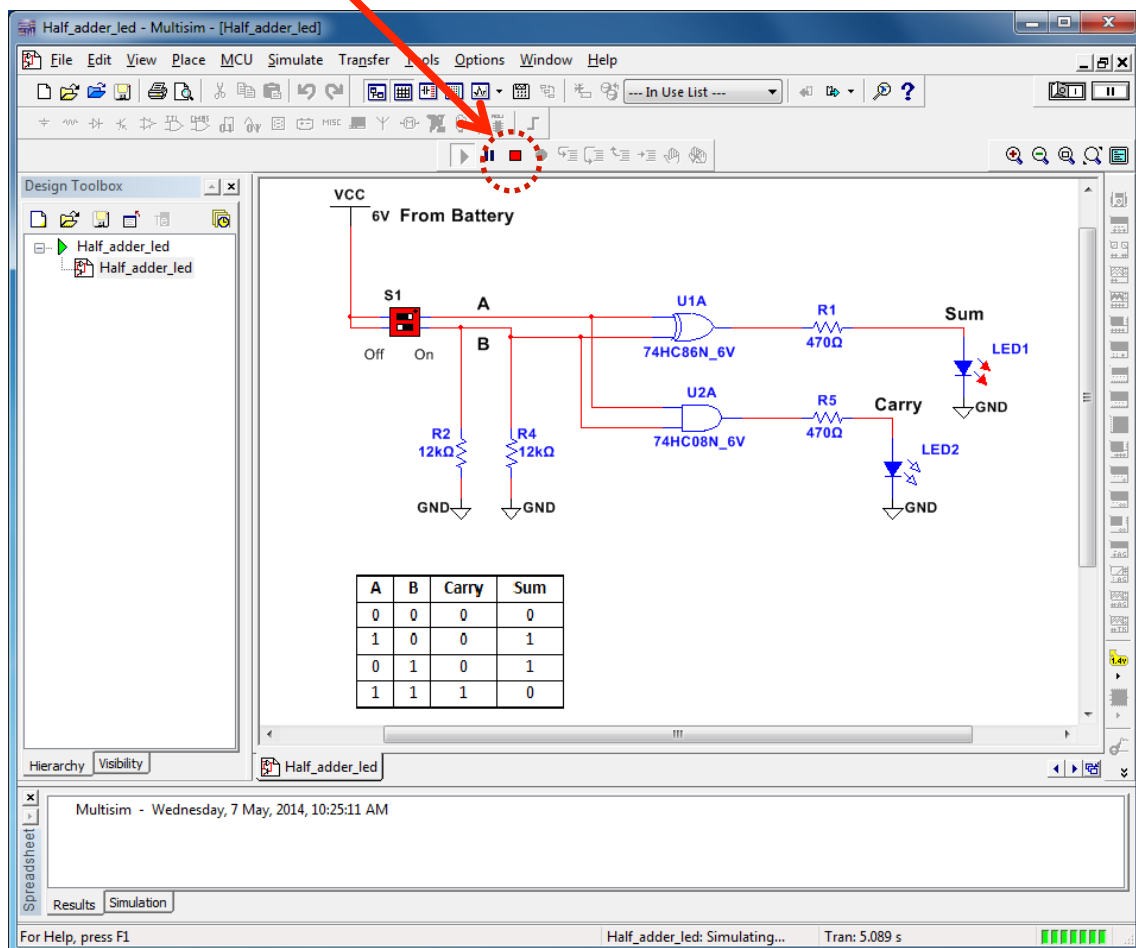


Figure 10: Simulation of the half-adder with A=1, B=0 where Sum=1, Carry=0 (LED1 in red means turn-on)

8. You can try to turn on the switch to verify the circuit as in Figure 10. You can stop the simulation by clicking on the Stop button.
9. Save and close the circuit.

Note: “From Battery”, “Off”, “On”, “Sum” and “Carry” are all written by using “Text”.

Transfer from Schematic to PCB

Objective:

Transfer the schematic designed in Simulation in Multisim to a useable PCB layout using **Ultiboard**.

Procedure

1. Open the saved file from the end of Simulation in Multisim.
2. Transfer the schematic design to **Ultiboard 12.0** by selecting **Transfer » Transfer to Ultiboard 12.0**.

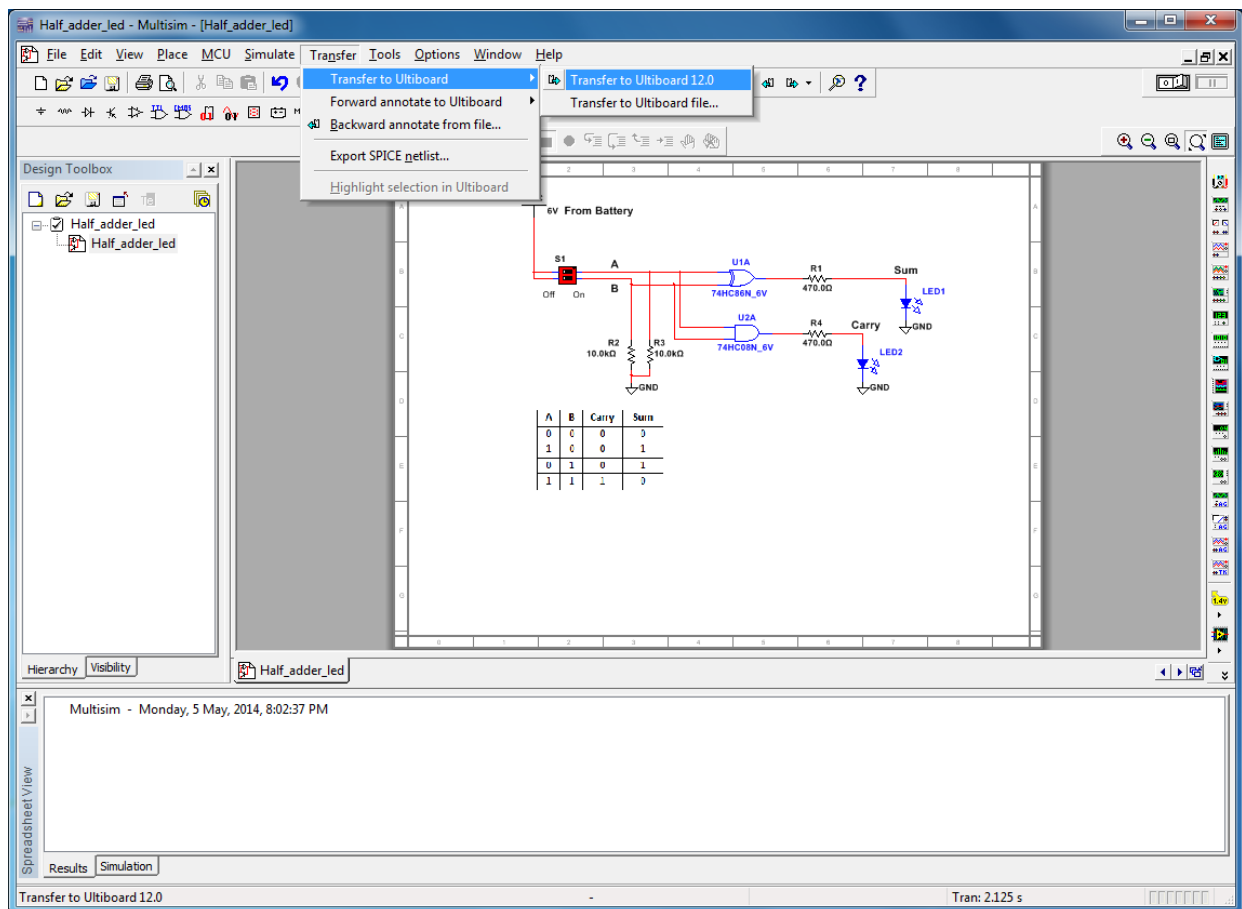


Figure 11: Transfer to Ultiboard

3. Use the **Save As** dialog box to save the new **Ultiboard** file to your preferred directory. Click **OK** on the message box that appears.
4. The program should look like Figure 12.

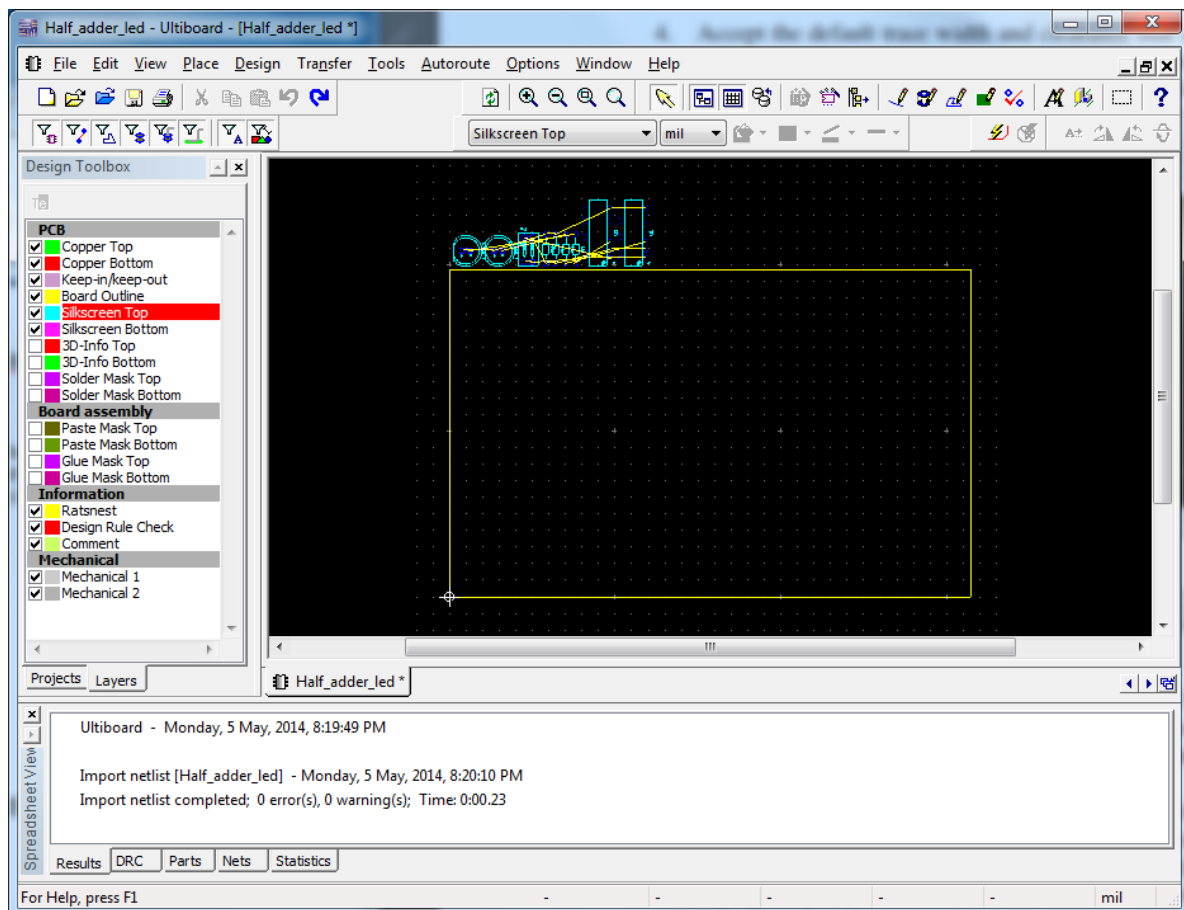


Figure 12: Ultiboard interface

- If the resistors are not printed in the Ultiboard like Figure 12, you have to change **Footprint** of the resistors as shown in the Red box of Figure 13.

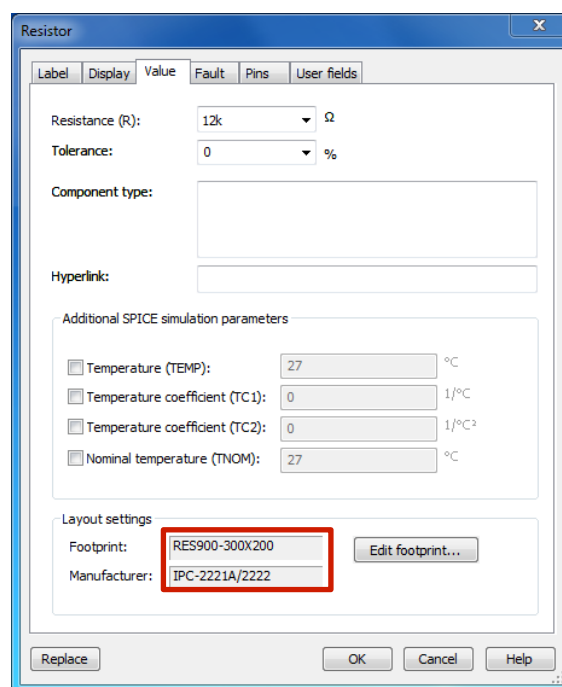


Figure 13: Footprint for Resistor

6. Select the **Layers** tab near the middle left of the screen. This lists the board layers and other useful elements of the board. Double-click **Board Outline** from the list to set this layer active.
7. Right-click an edge of the yellow rectangle and select **Properties** from the menu that appears. Under the **Rectangle** tab set the width to 2500 and the height to 2500 as shown in Figure 14. 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.

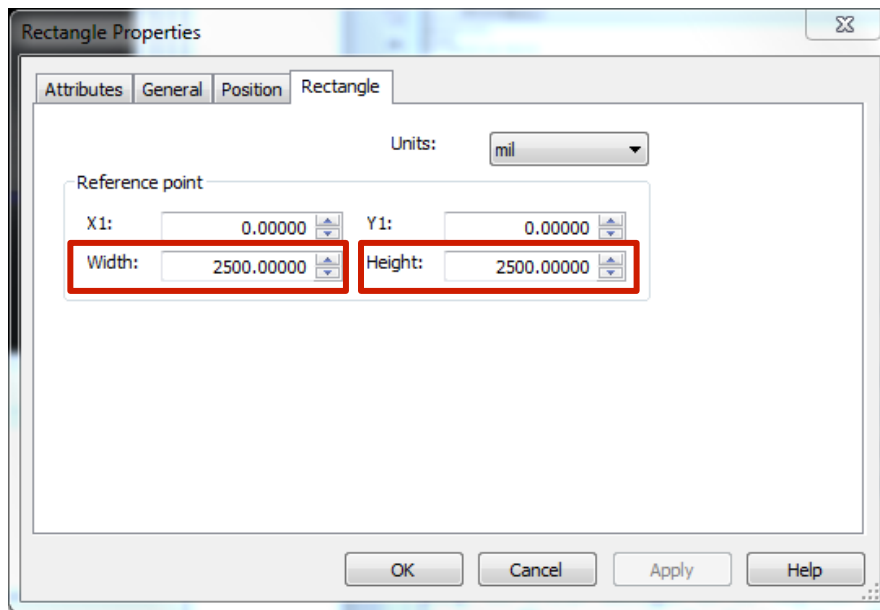


Figure 14: Board outline size

8. 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.
9. Begin placing the components inside the yellow rectangle. There are many different ways this board could be laid out and it takes practice to make efficient boards. Use Figure 15 as an example of where to put components, or feel free to create your own unique layout.

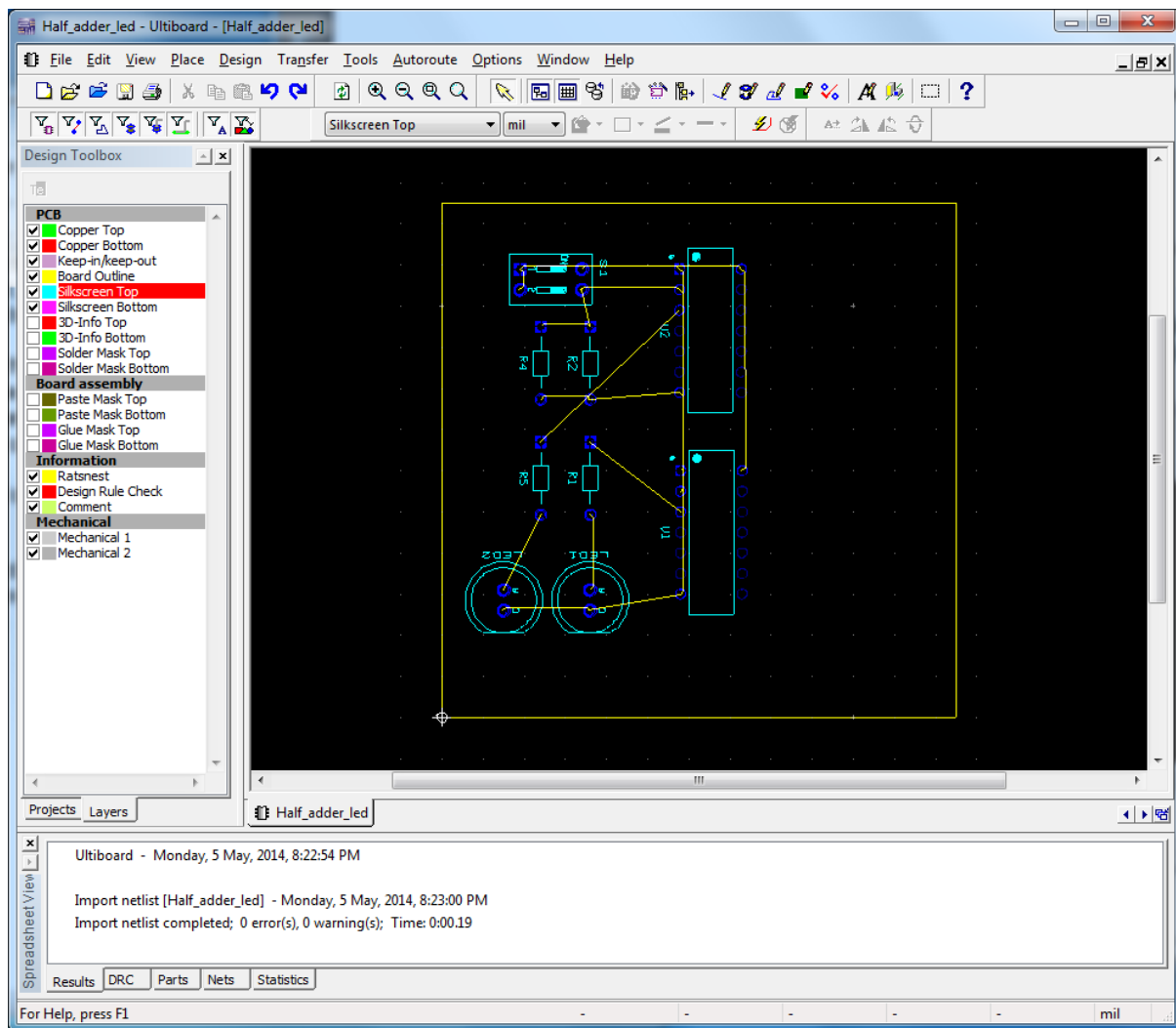


Figure 15: Placement of the components inside board area

10. The next step is setting up the routing parameters for laying out the copper trace. Go to the **Netlist Editor**, Figure 16 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 Top / 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.

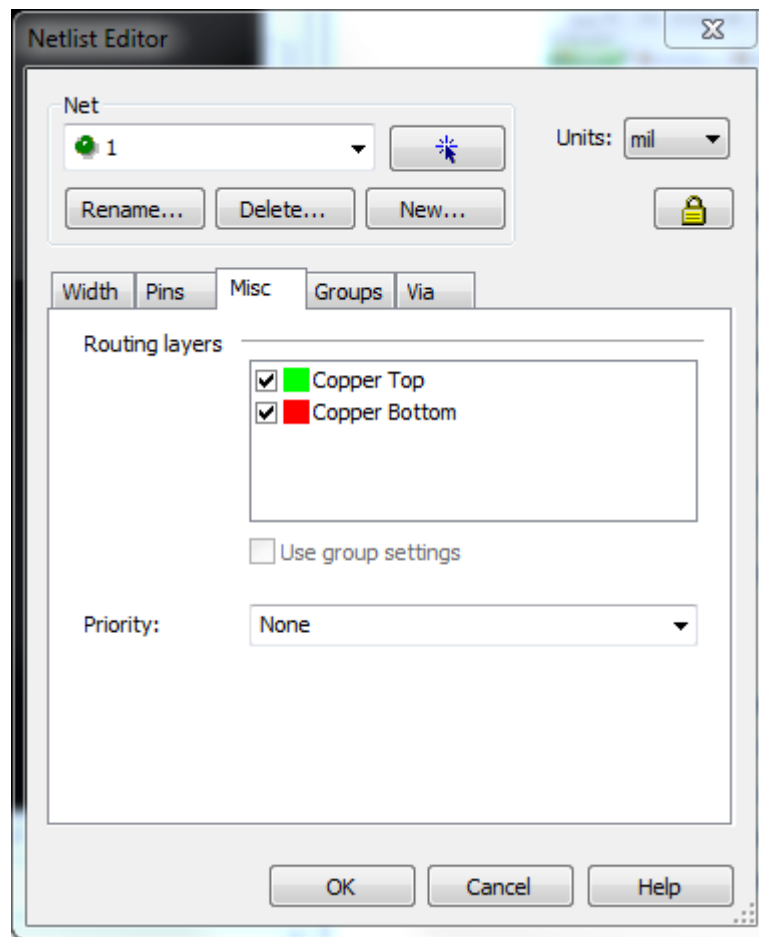
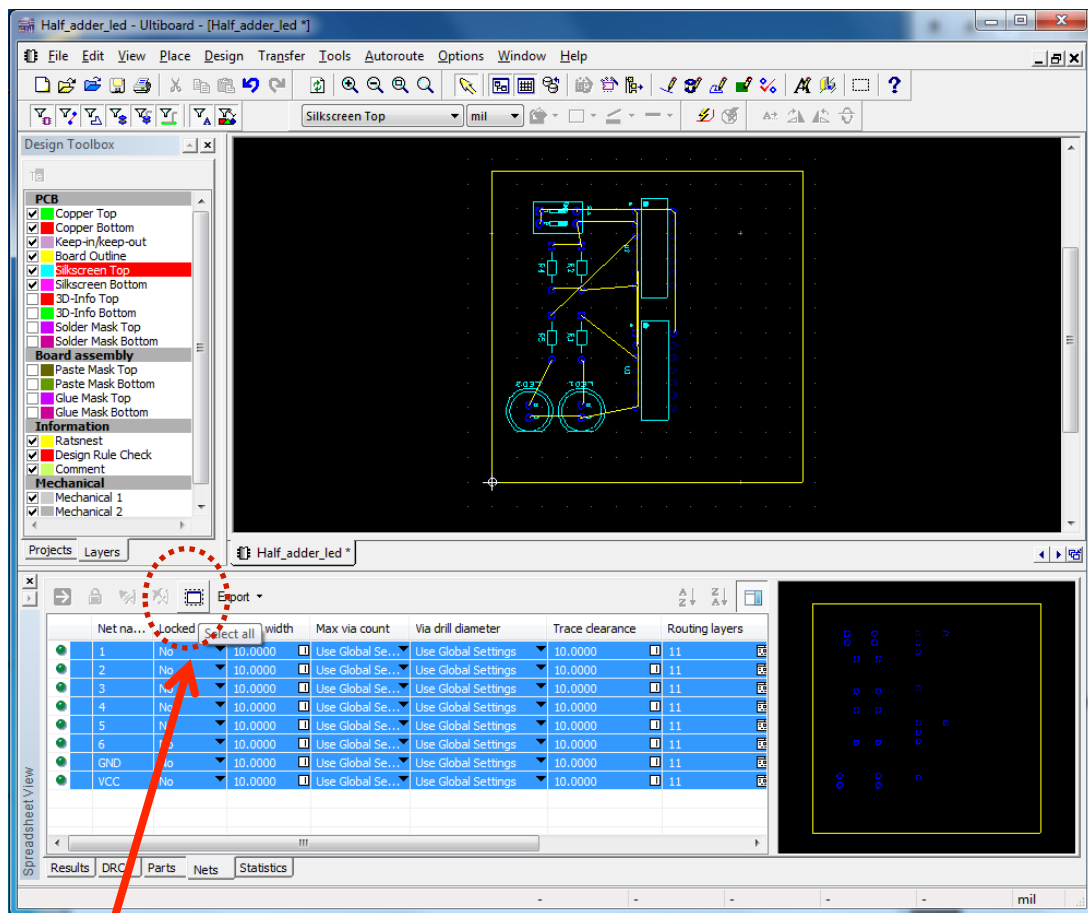


Figure 16: Netlist editor, selecting Routing layers

12. In this case, it is best if the **Autrouter** 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 on one of the rows and choose **Select All** from the menu that appears as shown in Figure 17. Click inside one of the cells in the **Routing Layers** column. Check the box next to **Copper Top/ Copper Bottom** in the window that appears and click **OK**, Figure 17.



Select All

Figure 17: Select all

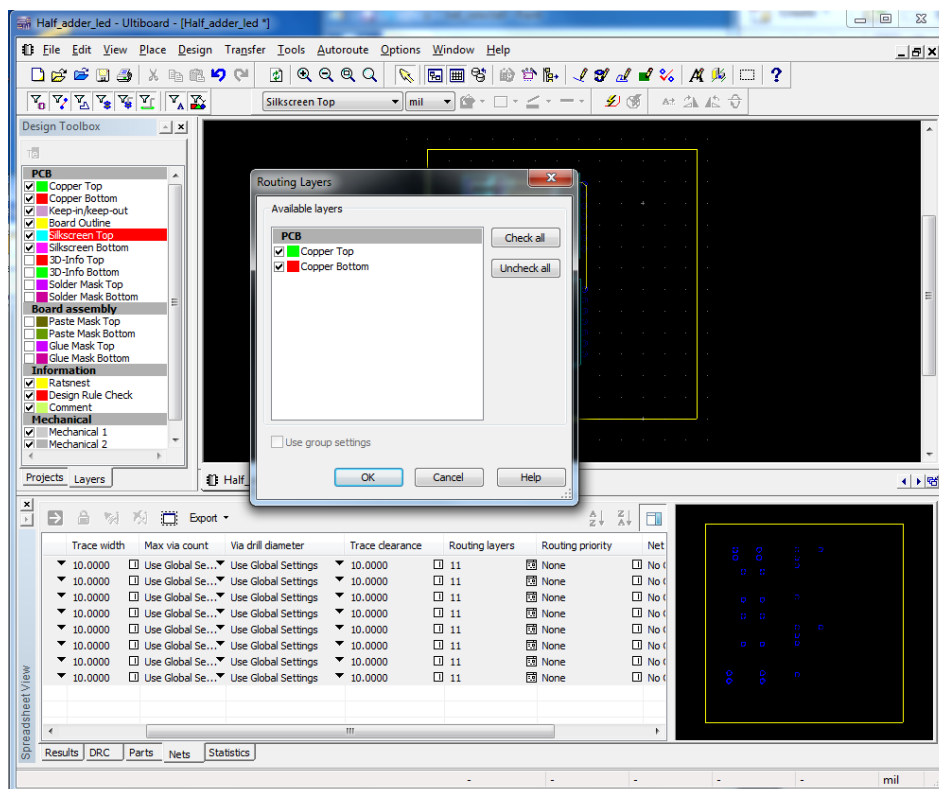


Figure 18: Select Routing layers

13. The board layout is now ready for the Autorouter. Start the Autorouter by selecting **Autoroute » Start/Resume Autorouting**. 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.
14. Figure 19 PCB after auto-routing shows an example of what the board from Figure 15 looks like after routing.
15. 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.
16. You should perform a design check to make sure no design-rule-check (DRC) and netlist error, **Design >> DRC and netlist check**. You should not have any error notification at the bottom of the Ultiboard interface in **Results** tab.
17. You should also perform a design check to make sure no connectivity's error, **Design >> Connectivity check**, select **all net**. You should not have any error notification at the bottom of the **Ultiboard** interface in **Results** tab.
18. Once you are satisfied with the traces on the board Save and Close the design.

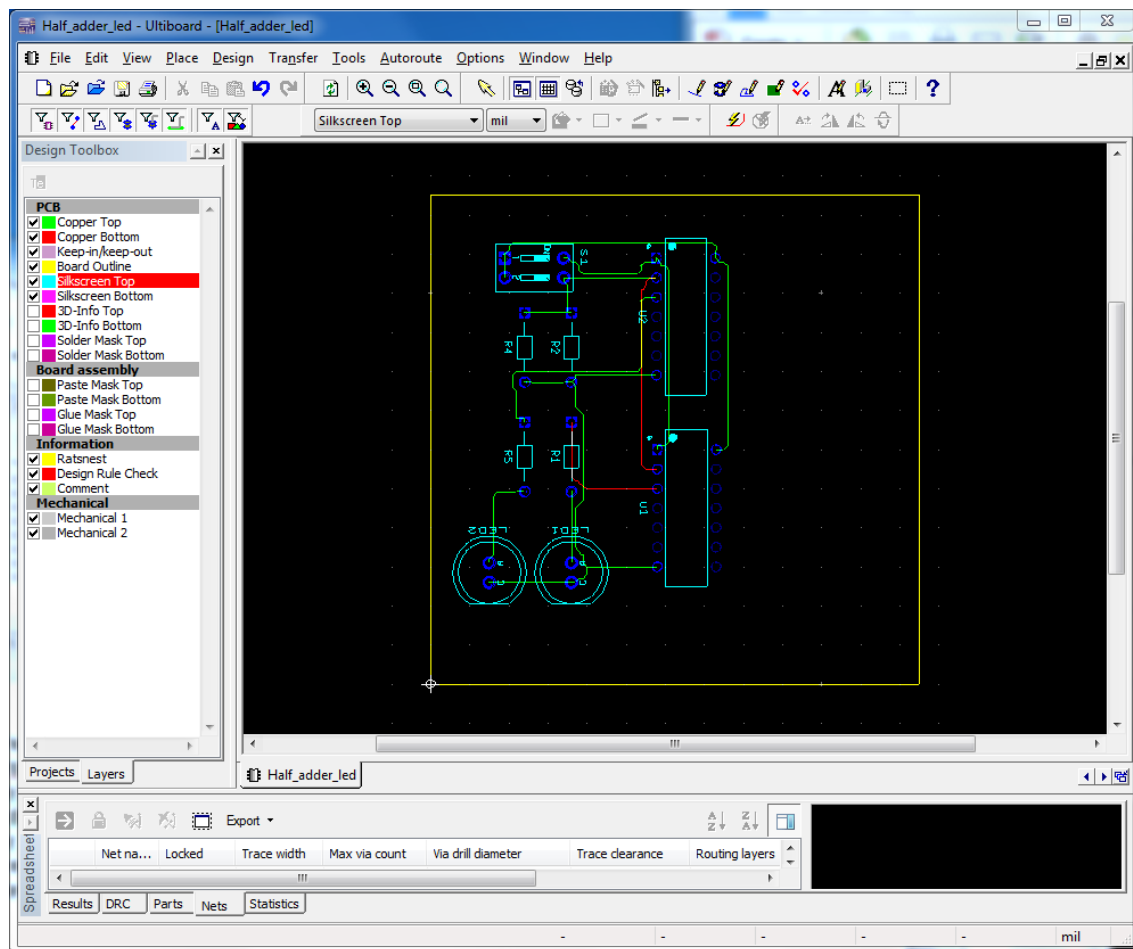


Figure 19 PCB after auto-routing