Lab 5: Drawing Circuits & Making a Printed Circuit Board
BMEn 2151 "Introductory Medical Device Prototyping"
## Prof. Steven S. Saliterman

5.1: Using Multisim and Ultiboard

Objective: Draw a familiar circuit in National Instruments Multisim and run a simulation. Next create a printed circuit with Ultiboard.

You can purchase Multisim online from National Instruments for your personal computer at a low cost as a student from *Studica.* The left-hand computer in the Electronic Prototyping Lab has Multisim on it.

Task

1. Draw this circuit in Multisim. Unlike the previous version of this binary counter, you are going to add a connector for the clock and power (HDR1X4).
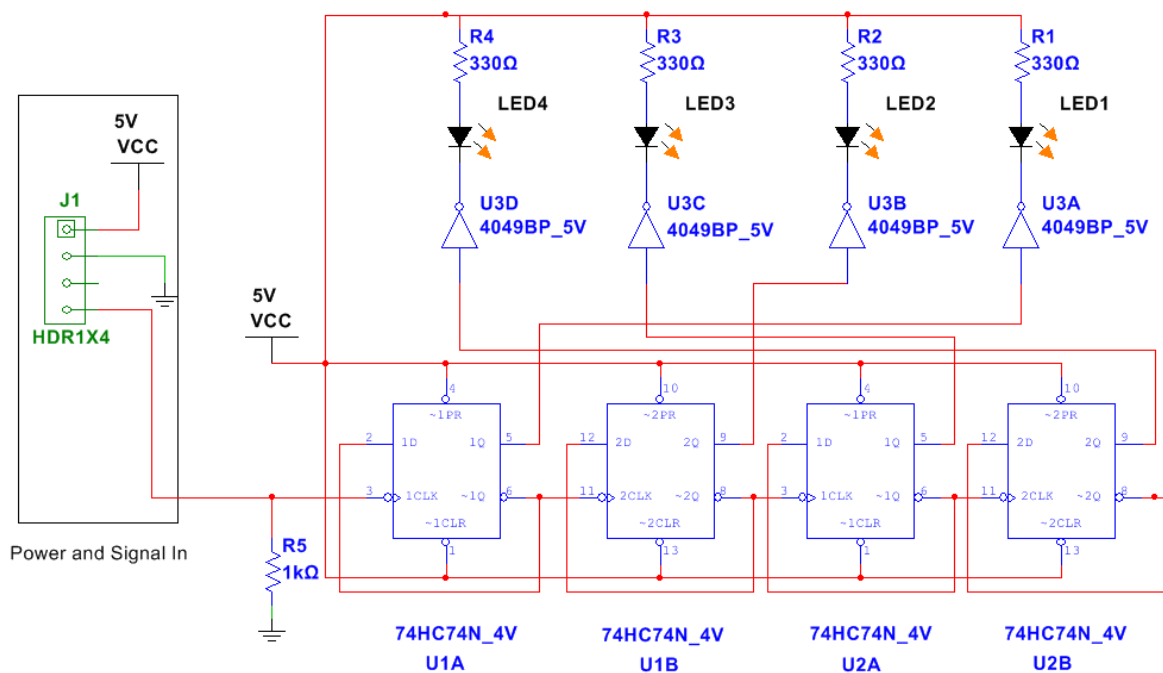

The resistors may initially have no manufacture's footprint for placement on a circuit board. Double clicking on the part opens its feature dialog. Under the "Value" tab at the bottom, "Edit footprint" is selected, and a suitable footprint chosen from the list available. If you were producing this board, you would want to be sure that each footprint matched the actual part you plan to use. Take a moment to inspect a part's features.
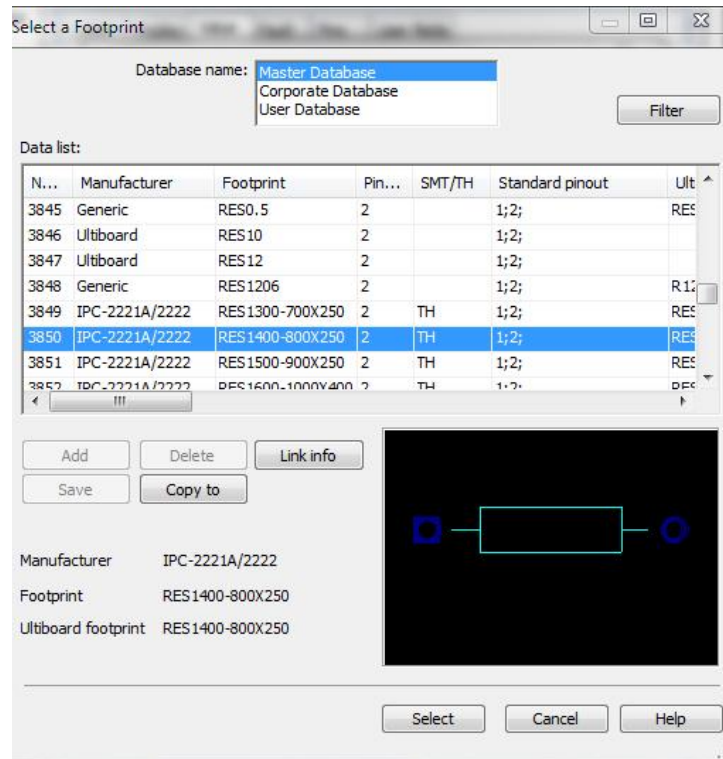
2. Next select "Transfer to Ultiboard" from the "Transfer menu." A net list will appear as shown below:
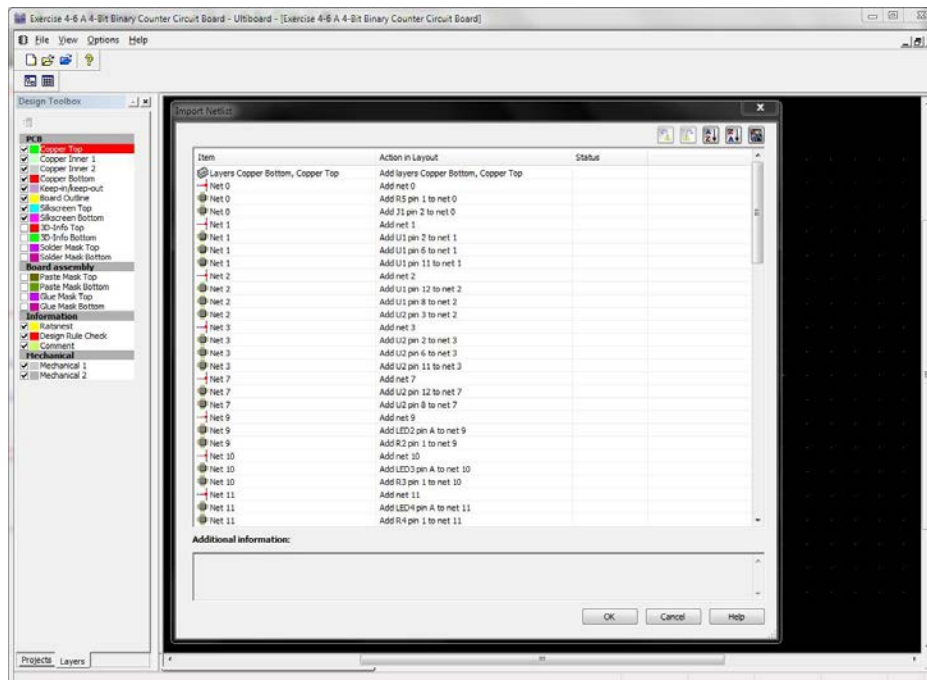
3. Select "OK" and the circuit board layout workspace will appear. You may need to select "3D View" and "Spreadsheet" under "View" menu (and slide the window to the right).
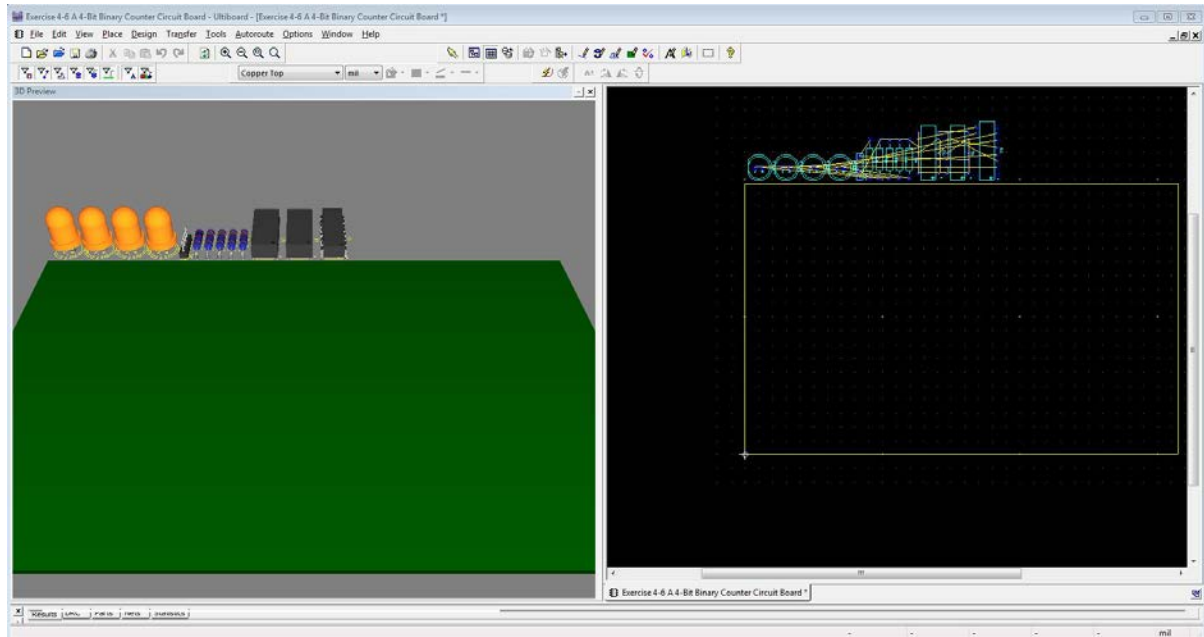
All of the parts are first seen on the top. Inside the workspace (right-hand side) drag each part into position as shown (there is no special layout).
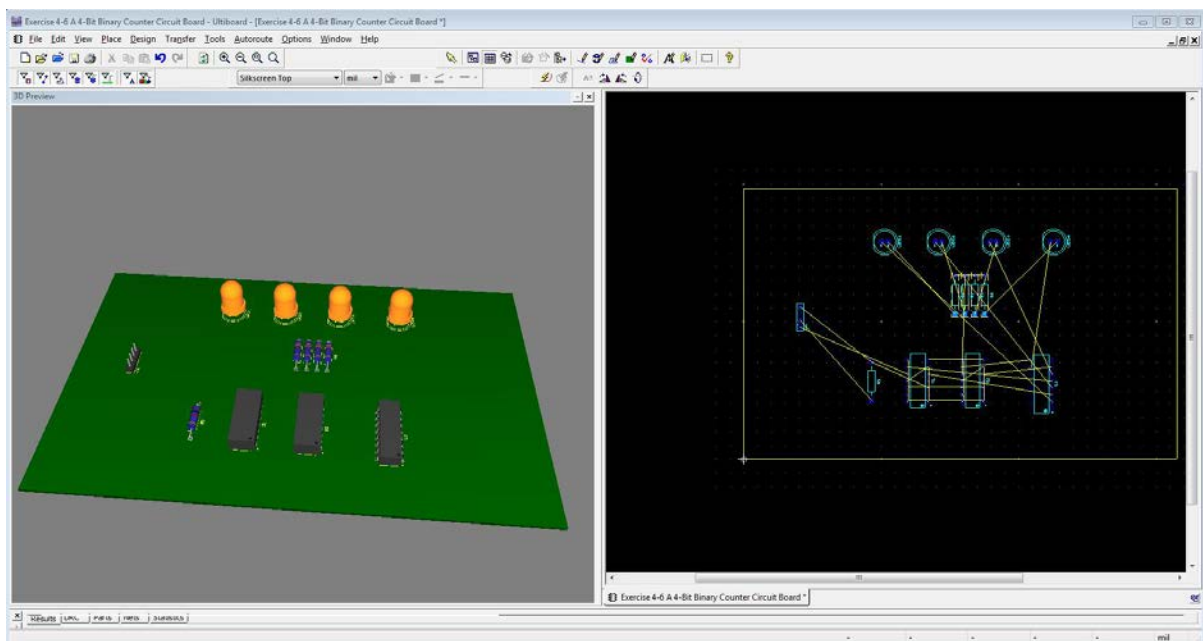
4. Select "Start autorouter" from the "Autorouter" menu. The software will automatically route each of the copper traces from pin to pin based on the netlist.
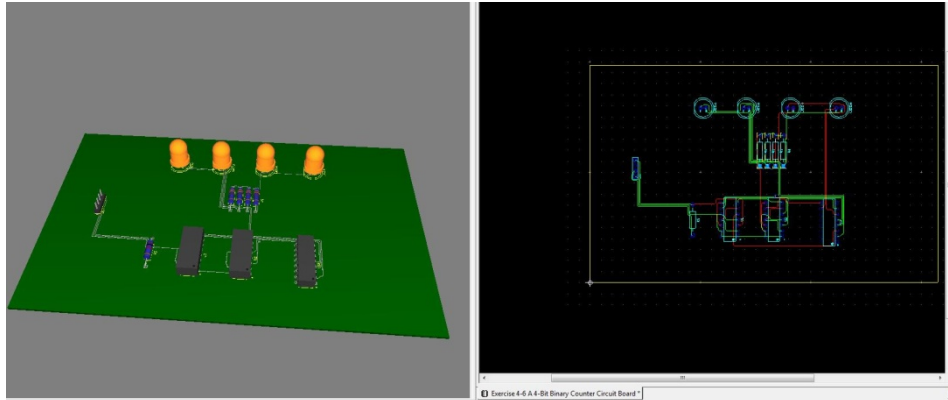
The board has now been populated and pins routed as specified in the netlist. You can use your mouse to rotate the board and visualize the traces underneath.

5. Create a JPEG (screen shot) of your circuit board in 3D and fully routed in the workspace as shown above. To do this, under "Tools" select the Capture Screen Area tool. Attach this to your *Labs 4-8 Workbook Assignments*.

Save your work and quit Ultiboard and Multisim.
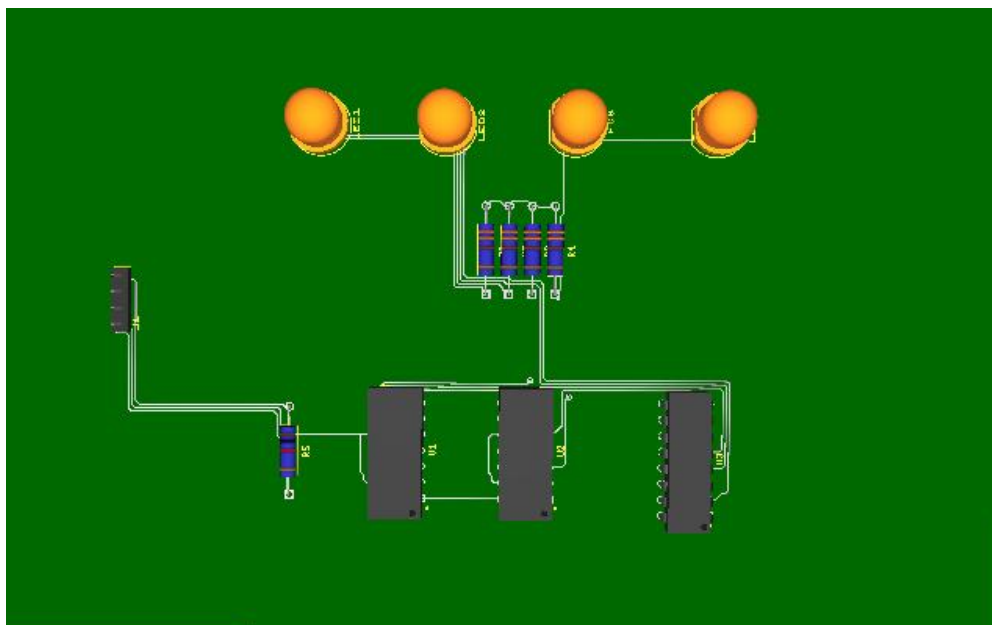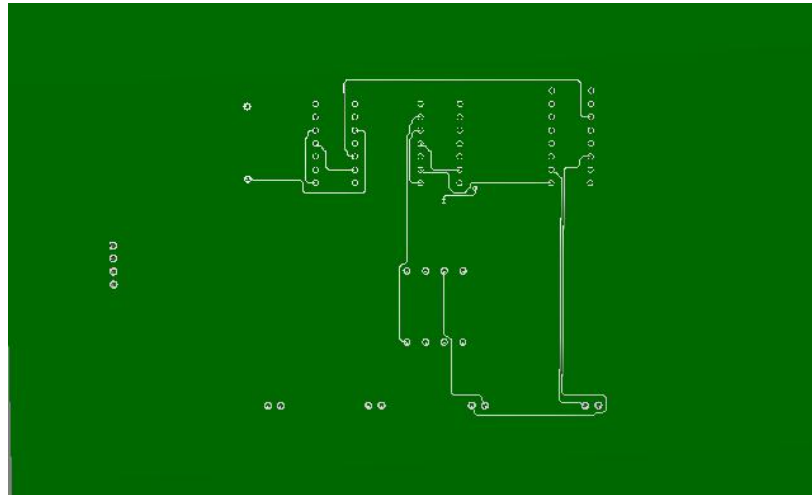
To have this board manufactured you must export the file with relevant information concerning the physical properties of the copper clad board, silk screen layer, and even the holes through the board (vias). The export dialog is as follows:
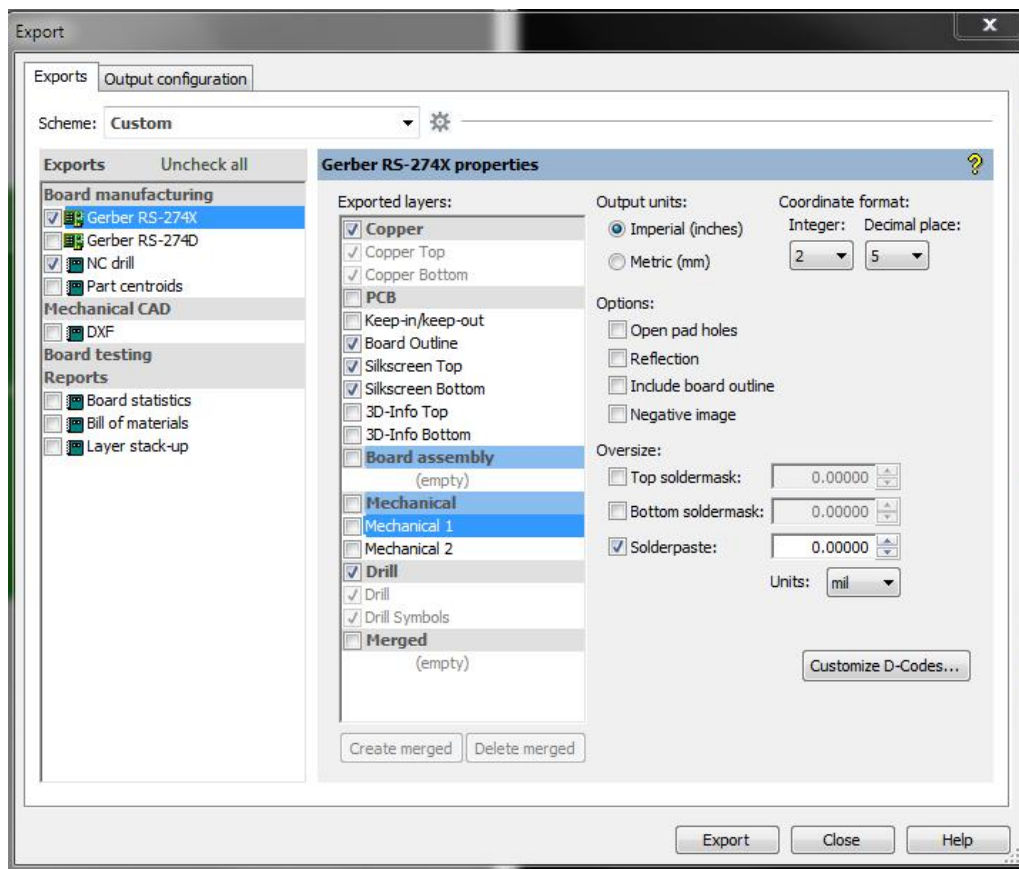
5.2 Use EasyEDA to Draw a Circuit, Create a Circuit Board, Make a Bill of Materials, and Determine the Cost of Production.

For this exercise you will draw and design a printed circuit board. This will consist of a top and bottom copper layer and top silk screen layer. It should be 1oz copper cladding. Start with the example circuit provided. You may then select your own circuit for design.

For those of you who wish to create your own design, start with something simple and build your complexity over time. You may optionally continue this exercise throughout the semester, and email questions to me as you progress.

Our lab session will consist of a tutorial on how to use EasyEDA. After the tutorial you will try a simple project during the lab session.

Task

1. Go to the EasyEDA website and register as a user. Your work will be saved on their cloud based system, and may be backed up on your computer. Create a new project titled: "My First Circuit Board." You will be creating a "schematic" and "circuit board" of an example circuit that I will distribute to you by email or post on our website. Save these individually to you project as simply "Drawing 1" and "Board 1."

Typically, as your schematic drawing evolves and you make corrections, you will find yourself frequently updating your circuit board. You can also look at 2D and 3D views of the board, and obtaining pricing information at any time.

This is an iterative process until you are satisfied with your circuit, obeyed all design rules, and made sure your copper trace widths and vias meet the current carrying requirements of the circuit. When obtaining a quote, you will specify the number of layers (accept the default), copper cladding thickness (1 or 2oz), color of the board (green, blue, black or red) and number of boards you wish to order.

2. Place your name in the title block of the schematic drawing, and email a pdf of this and your circuit board to me.

Using EasyEDA

From the EasyEDA homepage, select "EasyEDA Designer." Under File select "New Project", type your name as owner, and give it a title. Accept the default path to the file, and save. With the project file open and visible on the left, under File select "New Schematic." You now may enter your schematic drawing, using the user library or LCSC part library. Try to match the value and footprint of the part as close as possible to the one you want. Place the

parts on the drawing area, and move around as needed. Use the drawing tools to wire each component together. Look back at Lab 3 & 4 to see how I drew the circuits for the exercises.

The best way to learn how to use the software is to simple experiment with it, going through the menu items and drawing tools.

When ready, under Design, select Convert to PCB. As this juncture you review the netlist, and will specify the board outline. You then pick and place each component on the board, and select autoroute. In the future, after you make revisions to the schematic, under File, select Update Circuit Board rather than Concert to PCB. Once your board is autorouted you can look a 2D or 3D view.

When all done, under fabrication, select BOM (bill of materials) to see the parts you are using. Next under Fabrication select PCB Order, and work through the steps of obtaining a quote (without actually placing an order!).

About Simulation

The EasyEDA has the ability to simulate a circuit as we did with Multisim. I do not recommend doing this. Every component you use must be from a specific manufacturer and have an accompanying p-spice drawing (a commonly used method for creating a part used in simulations). In contrast Multisim has a very large generic part library without needing to identify a manufacture's p-spice drawing. The simulation is also much more robust and easier to use. Therefor I recommend first drawing and simulating in Multisim, then, rather than using Ultiboard, use EasyEDA redraw your circuit, and make your circuit board. EasyEDA is more powerful for circuit board design, easy to use, and gives you pricing far better than the choices through other vendors.

End of Drawing Circuits & Making a Printed Circuit Board