Exercise 3 - PCB Design

Objectives

- Learn how to use EAGLE for a basic PCB layout (including routing traces, vias and ground planes)
- Learn how to run a Design Rule Check to verify that the layout matches the shop’s requirements.

Exercise

Step 1 - Open lpc_boilerplate.brd

1. Open EAGLE from the Start menu
2. From the left pane, select Projects ece395_projects lpc_blinky lpc_boilerplate.brd

You will see that many of the components have already been placed within the rectangular box that defines the edges of the board. As you start to route the connections between the components, you may want to move/rotate some components for easier routing and a cleaner board layout.

Step 2 - Set the grid display

1. Click on the button from the toolbar to open the Grid settings
2. Configure the grid settings as follows:
   1. Display: On
   2. Size: 20 mil
   3. Multiple: 1
   4. Alt: 10 mil

Step 3 - Create a GND plane

Step 4 - Routing traces and Vias

- Routing
- Vias

Step 5 - Run Design Rule Check

To-Do Before Next Lab

Additional Resources
Step 3 - Create a GND plane

1. Go to the ECE Electronics Services Shop’s website and navigate to Design Requirements for our PCBs. Ground Planes or Fill Areas. Read this section and find the recommended value for the “Isolate” parameter.

2. In EAGLE, select the Polygon tool from the left toolbar.

3. From the top toolbar, ensure that the Bottom layer of the PCB is selected.

4. Also from the top toolbar, change the Isolate parameter to the value recommended on the ECE Services Shop’s website.

5. Draw the Polygon around the edge of the board. Start by clicking on one corner, then click the diagonally opposite corner and click again on the first corner. This should complete the rectangular polygon around the edge of the board. You should now see a dashed-blue line along the edges.
6. Now select the Name tool from the left toolbar. Then anywhere click on the dashed blue line. In the dialog box that appears, type the name "GND". Then click OK. The polygon is now a Ground plane.

**Step 4 - Routing traces and Vias**

Notice the greenish-yellow lines running between the components' pins. These lines indicate connections between pins as defined on the schematic.

**Routing**

1. Click the Ratsnest button from the left toolbar. You will notice that the GND plane has been filled-in and some of the greenish-yellow connection lines have disappeared. The lines corresponding to GND connections disappeared as they no longer need to be routed as traces (the GND plane has made these connections).

   The filled-in GND plane may make it harder to see connections between components. To address this, select the Polygon tool again, right click on the edge of the board, and select Properties. Under Polygon Polygon Pour, select "cutout".
2. Select the Route tool from the left toolbar.

3. From the top toolbar, change the Wire Bend Style to Style 1 or 3.

4. Change the Width to 24 (mil) (see the trace width requirement on the ECE Services Shop Website).

5. Choose a component and start routing! When you click on any component's pin, move the cursor around and you will see how the trace and the yellow line follow the cursor. Follow the yellow line to the other component's pin to complete a trace route. You may need to use the top and bottom layers to complete the board layout. The layer can be changed from the top toolbar.

To remove a trace (e.g. to re-route or make a correction), use the Ripup Tool.

**Vias**

Vias are through-hole connections between layers of the PCB. Sometimes, to you may need vias to complete routing all the connections.

Follow these steps if you need to use vias:

1. Use the Route tool and start the trace as usual by clicking on a component's pin.
2. Move the cursor to the location where you want a via and click the left mouse button to draw the trace up to that point.
3. Then from the top toolbar, change the board layer.
4. Complete routing as usual.
5. When the trace has been routed to its destination pin, EAGLE will automatically create a via where the trace changed layers.

If you are having the PCB manufactured by the ECE Electronics Services Shop, you want to minimize the use of vias. Read their note about vias.

**Step 5 - Run Design Rule Check**

When you have completed routing all connections, the board design must be verified to ensure that it meets the requirements of the PCB manufacturer. For this exercise, you will check that your board meets the requirements defined by the ECE Electronics Services Shop.

1. Go to the ECE Electronics Services Shop's website and download the .zip file with the design rules (Design Requirements for our PCBs Autorouter Parameters and Design Rules). Extract the .dru file from the downloaded zip folder.
2. In EAGLE, select the DRC button from the left toolbar
3. From the File tab, click on "Load..." and browse to the extracted file
4. Click "Check"
5. If there are no errors, the status bar at the bottom of the window will show DRC: No errors.

You MUST NOT use the autorouter to complete this exercise!

To-Do Before Next Lab

- Complete the PCB layout design by routing all the traces.
- Show the TA that your design passes the DRC with the .dru file from the ECE Electronics Services Shop.

Additional Resources

Sparkfun's EAGLE: Board Layout Tutorial