Turning an EAGLE schematic into a PCB

Andrew Haumersen
ECE 480
Design Team 5

Application Note
11/7/13
**Executive Summary:**

This application note demonstrates how to convert a preexisting EAGLE schematic into PCB ready for submission to a company for printing. It includes creation, layout, routing, sizing, and modification as well as a few helpful tips to keep routing neat.

**Introduction:**

For our project team 5 was tasked with creating a breakout board for Chrysler. We decided to do this using EAGLE to create a schematic and then turn the schematic into a PCB board. The following steps demonstrate how this was done for our project.

**Step One: Converting Schematic to Board Layout**

Before you can create a PCB layout you must first have a complete schematic outlined in EAGLE. For our project below the schematic is very large and complex.
Once you have all of your devices modeled and laid out on the schematic it is very important to make sure that every wire is labeled correctly and has a proper connection. This next image is a close up to display every connection has a proper label.

Once you have your schematic all ready to convert to PCB. As displayed below you access file>switch to board, it will then prompt to create a new board from the schematic for you, click “yes”.
After this is completed the board window will appear and you will have a blank PCB with all of the components outside and ready to be placed as displayed below.
Step Two: Placing objects onto PCB

The next step is to place all of these components within the blank PCB in the location you want them. This task can be very large and take a lot of time depending on the complexity of your circuit. In order to grab objects and move them to the right location you must click on the move command and then select the desired object and move it into the PCB. As displayed below all of the labeled connections will have “air wires” that stay connected no matter where the object is moved. A useful command to help keeps these under control is tools>RATSNEST, this will move the air wires to the most direct route, cleaning up the PCB.
It is very important to keep all of the objects neat and organized to help with routing the traces in the next step. Below is an image of our PCB with all of the components placed in their final locations and all of the air wires running between them.

![PCB Image]

**Step 3: Routing traces for PCB**

In order to start routing traces you must select the route command and then select the location that you want to start your trace from. Remember that you can only route traces where air wires are already located. You can also use multiple layers for different traces by selecting the layers command and selecting a different layer to run the trace, each layer will have its own color which can be changed using the change button. This is the most important step and the most time consuming in creating a PCB. It is important to plan ahead and lay out your traces so that you do not get boxed in or have to cross traces. The image below is our schematic after every trace has been completed.
Once you have completed all of your traces you want to use the DRC command to check for errors. If your circuit is correct you will have a message saying “0 errors found” displayed and you are ready to send your PCB file to get printed!