

Turning an EAGLE schematic into a PCB

Andrew Haumersen

ECE 480

Design Team 5

Application Note

11/7/13

Executive Summary:

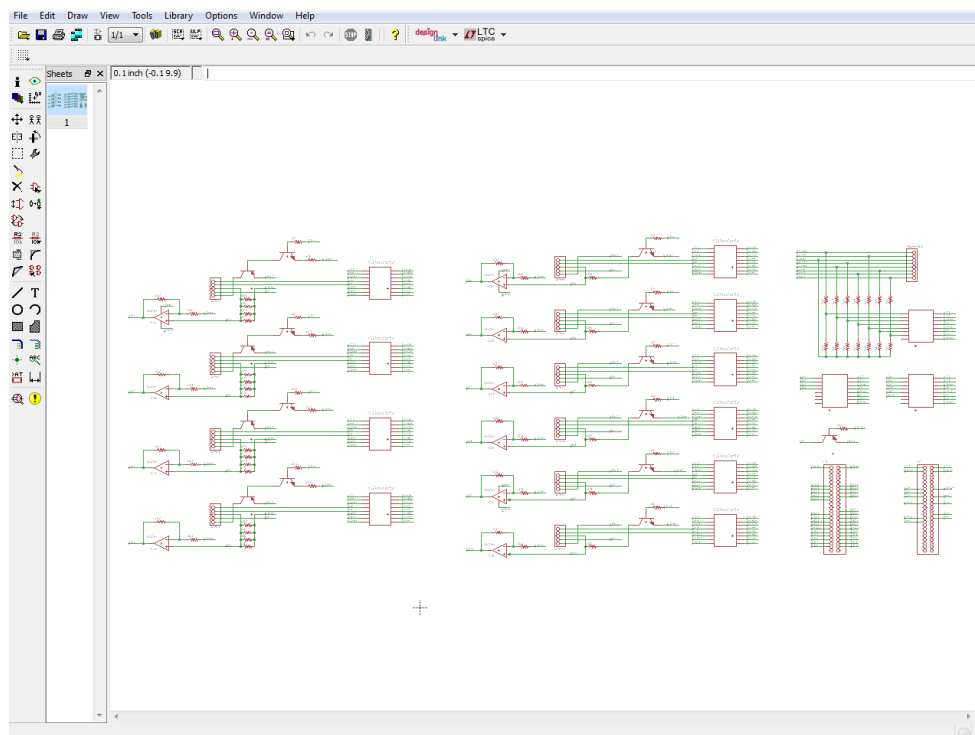
This application note demonstrates how to convert a preexisting EAGLE schematic into PCB ready for submission to a company for printing. It includes creation, layout, routing, sizing, and modification as well as a few helpful tips to keep routing neat.

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.

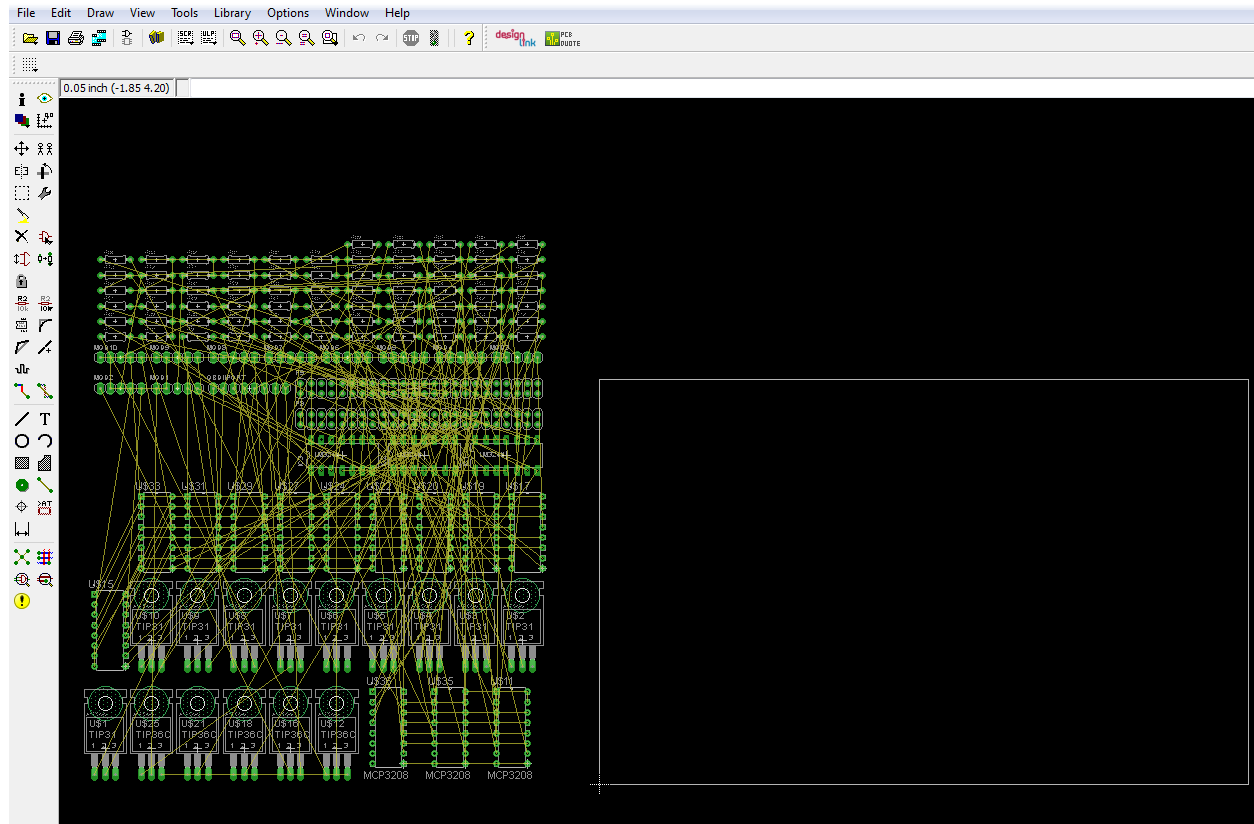
Step One: Converting Schematic to Board Layout

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.



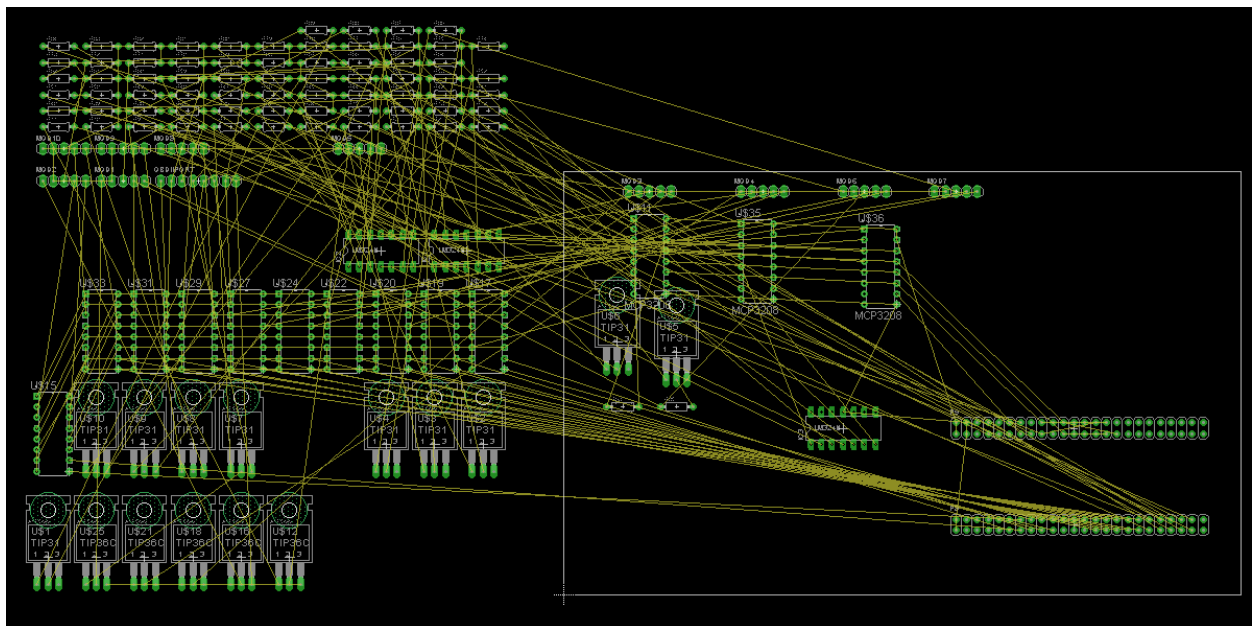
[illegible]

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.

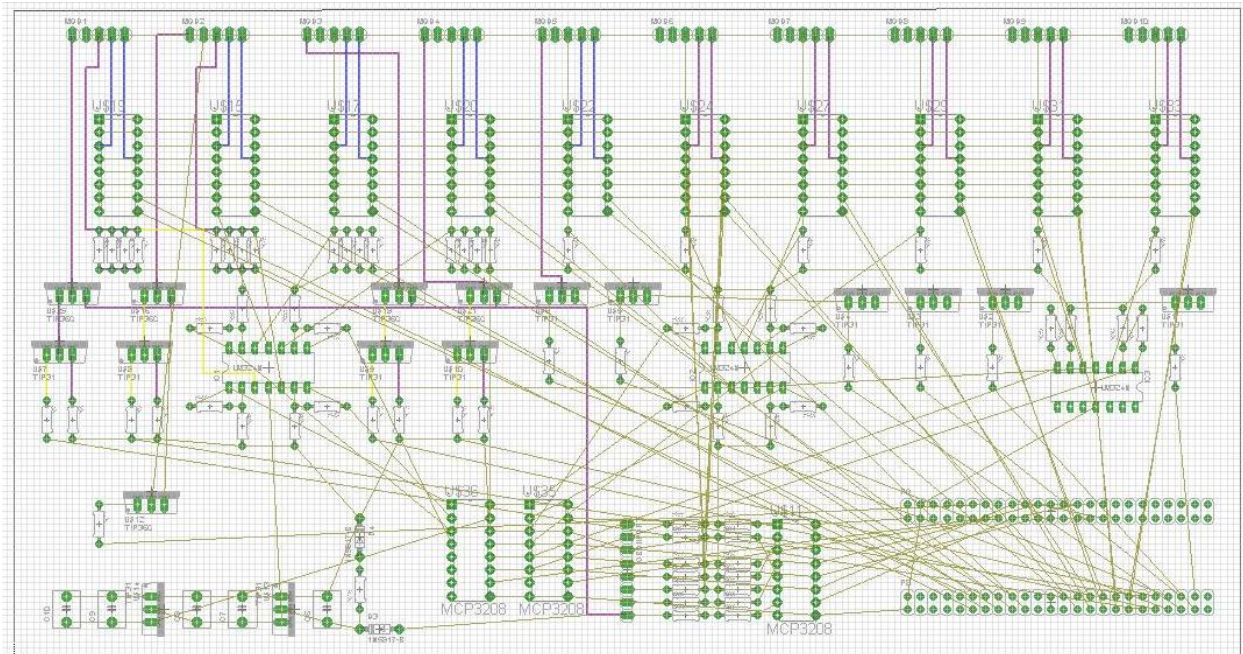


Step Two: Placing objects onto PCB

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 keep these under control is `tools>RATSNEST`, this will move the air wires to the most direct route, cleaning up the PCB.

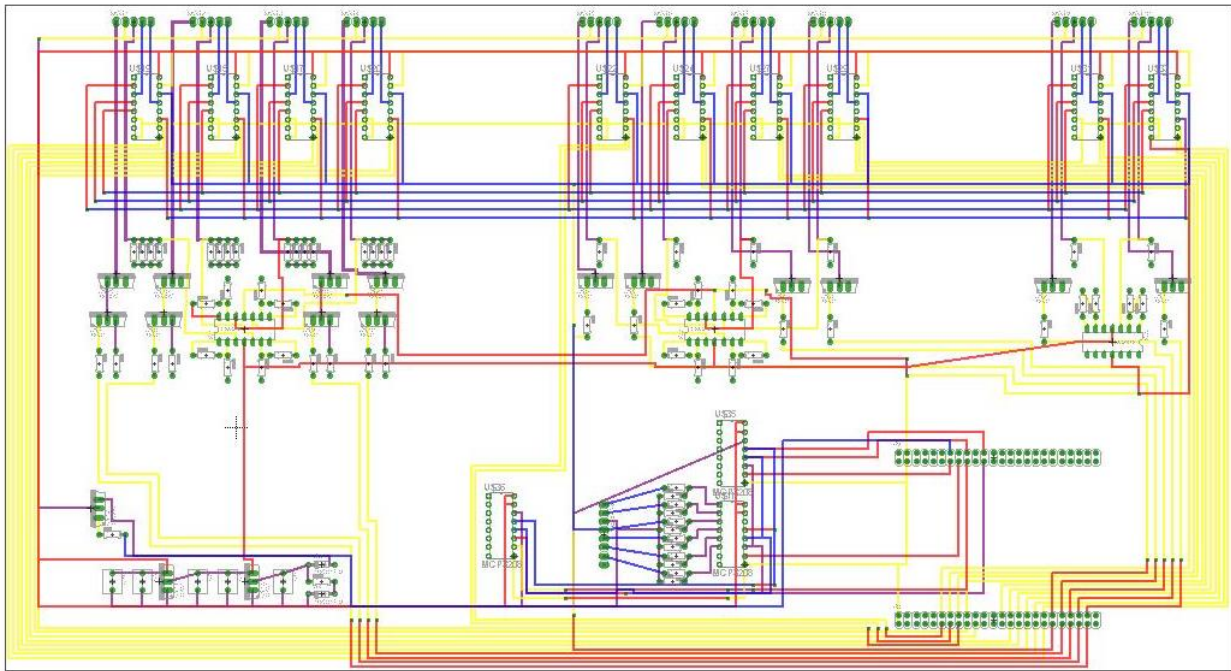


It is very important to keep all of the objects neat and organized to help with routing the traces in the next step. Below is an image of our PCB with all of the components placed in their final locations and all of the air wires running between them.



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!