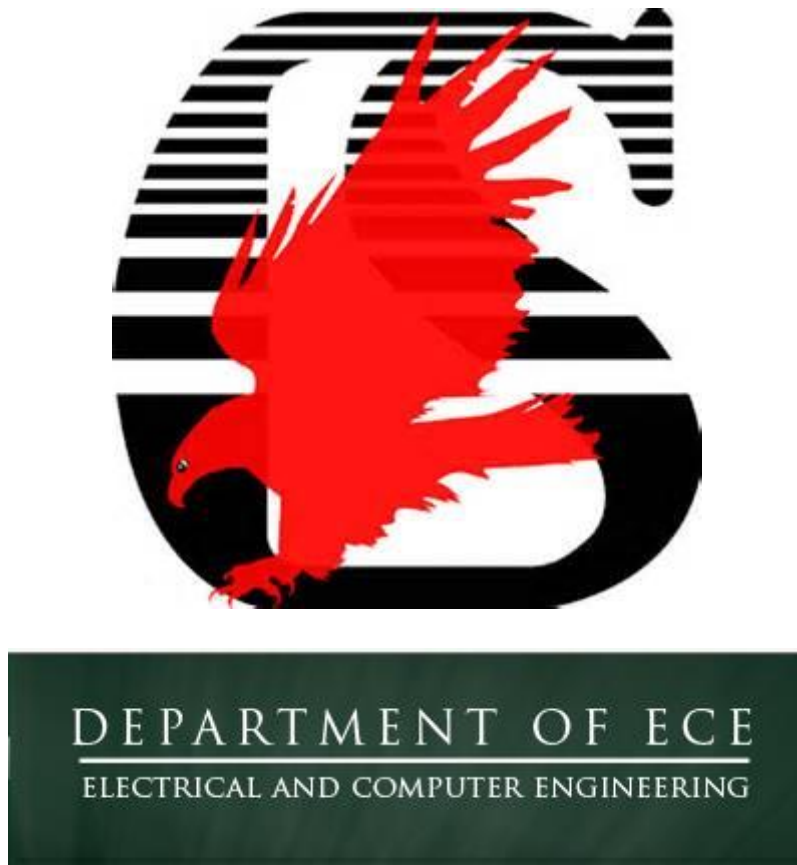


Creating a Board File from a Schematic File in Eagle

7.3.0 (for fabrication in the Michigan State ECE shop)



Written by: Alex Sklar

Keywords: Eagle, Michigan State ECE Shop, Board file, Bass Boost Headphone Amp

Executive summary: The content contained in the following application note demonstrates the process of creating a Printed circuit board from a schematic file in Eagle. It is assumed that the user is familiar with creating schematic files and all of the components needed for the project. The tutorial will use an example circuit to demonstrate how to transfer schematic files, route traces, and finalize the board for fabrication. This procedure is optimized for designing printed circuit boards in the Michigan State University ECE shop but a similar process may be used for other fabrication methods.

Table of Contents

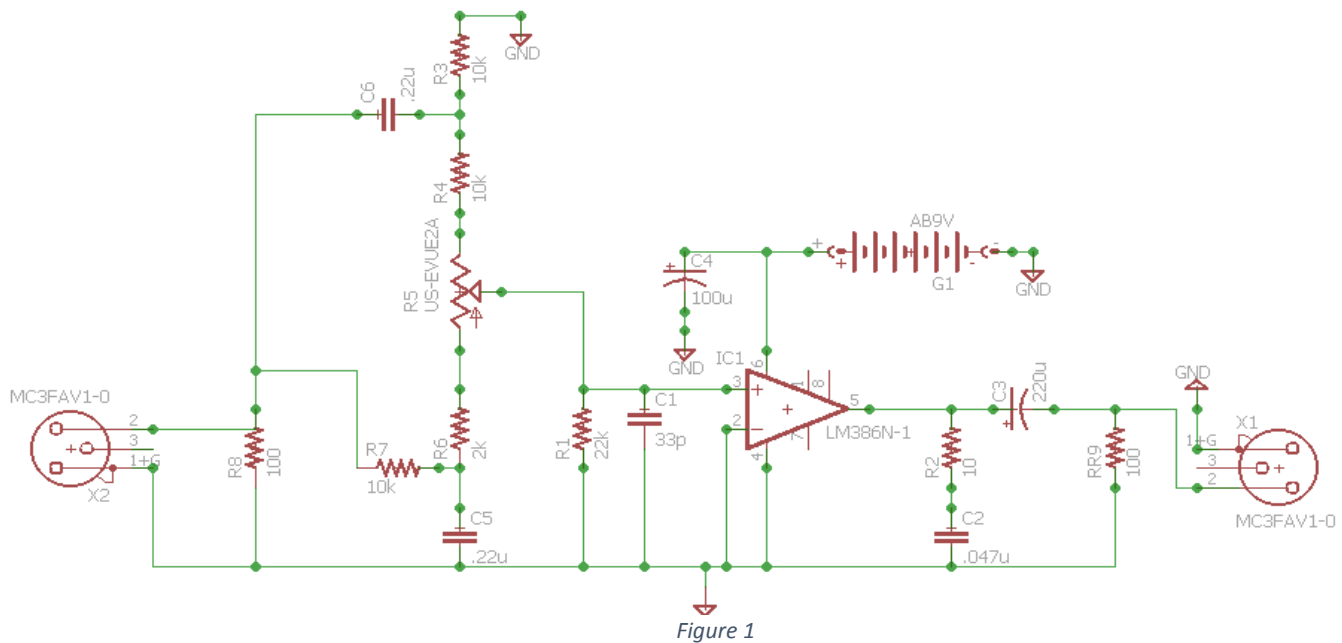
- 4-** Transferring the Schematic file
- 6-** Laying out the board
- 9-** Routing the traces manually
- 12-** Routing traces with the Autoroute tool
- 13-** Finalizing the board
- 16-** Exporting the board for production
- 17-** Conclusion
- 18-** References

Introduction: When it comes to prototyping a circuit or finalizing a design, a printed circuit board provides a more robust alternative to breadboards. Unlike breadboards, components are soldered onto a premade printed circuit board which makes the design more durable and easier to mass produce. Another advantage that printed circuit boards have, is the ability to adjust the size of the traces. By adjusting the trace width and material, it is possible to draw much higher amounts of current on the printed circuit board than is possible on a breadboard. While printed circuit boards may vary in construction material, the process of creating these boards is generally the same. Eagle is a software package that allows users to create a printed circuit board and create various types of files for fabrication. Eagle uses two main steps to complete this process. The first step is to create a schematic file to import all of the components that are used and make the electrical connections. This file is then referenced to create a board file which is used to layout the physical properties of the board. This application note uses a circuit from “Make” [1] found at <http://makezine.com/projects/bass-bump-headphone-amp/> , to create the board file for a simple bass bump headphone amp.

Objective: While it may be simple to create a schematic file in Eagle, it becomes a task to transform this into the physical layout that will be fabricated. Unlike the schematic file, physical issues come into play when creating the board file. This includes choosing the correct trace width for the applicable current, routing traces on the correct layers to avoid overlapping, and using a ground plane to ground the proper connections. These issues may be new to some who have little experience designing printed circuit boards, but the step by step tutorial for the bass bump headphone amp will show the process for creating a printed circuit board for any circuit.

Transferring the schematic file

1. Start by reviewing the project as shown in “Make” Bass Bump Headphone Amp [1]
2. Open Eagle and create a schematic file as shown in **Figure 1** below. **Table 1** includes the list of components used in the example schematic file.



Part	Value	Device	Package	Description
C1	33p	C2,5-3	C2.5-3	CAPACITOR
C2	.047u	C2,5-3	C2.5-3	CAPACITOR
C3	220u	CPOL-USTT5D9	TT5D9	POLARIZED CAPACITOR, American symbol
C4	100u	CPOL-USTT5D9	TT5D9	POLARIZED CAPACITOR, American symbol
C5	.22u	C2,5-3	C2.5-3	CAPACITOR
C6	.22u	C2,5-3	C2.5-3	CAPACITOR
G1	AB9V	AB9V	AB9V	9-V BATTERY CLIP
IC1	LM386N-1	LM386N-1	DIL08	Low Voltage Audio Power Amplifier
R1	22k	R-US_0204/2V	0204V	RESISTOR, American symbol
R2	10	R-US_0204/2V	0204V	RESISTOR, American symbol
R3	10k	R-US_0204/2V	0204V	RESISTOR, American symbol
R4	10k	R-US_0204/2V	0204V	RESISTOR, American symbol
R5	US-EVUE2A	US-EVUE2A	EVUE	9 mm Square Rotary Potentiometers
R6	2k	R-US_0204/2V	0204V	RESISTOR, American symbol
R7	10k	R-US_0204/2V	0204V	RESISTOR, American symbol
R8	100	R-US_0204/2V	0204V	RESISTOR, American symbol
RR9	100	R-US_0204/2V	0204V	RESISTOR, American symbol
X1	MC3FAV1-0	MC3FAV1-0	NC3FAV1-0	Neutrik Audio Connector XLR SERIES
X2	MC3FAV1-0	MC3FAV1-0	NC3FAV1-0	Neutrik Audio Connector XLR SERIES

Table 1

3. click on the **File** tab at the top left of the screen and click on **switch to board**
4. A screen should pop up that looks similar to the screen in **figure 2**

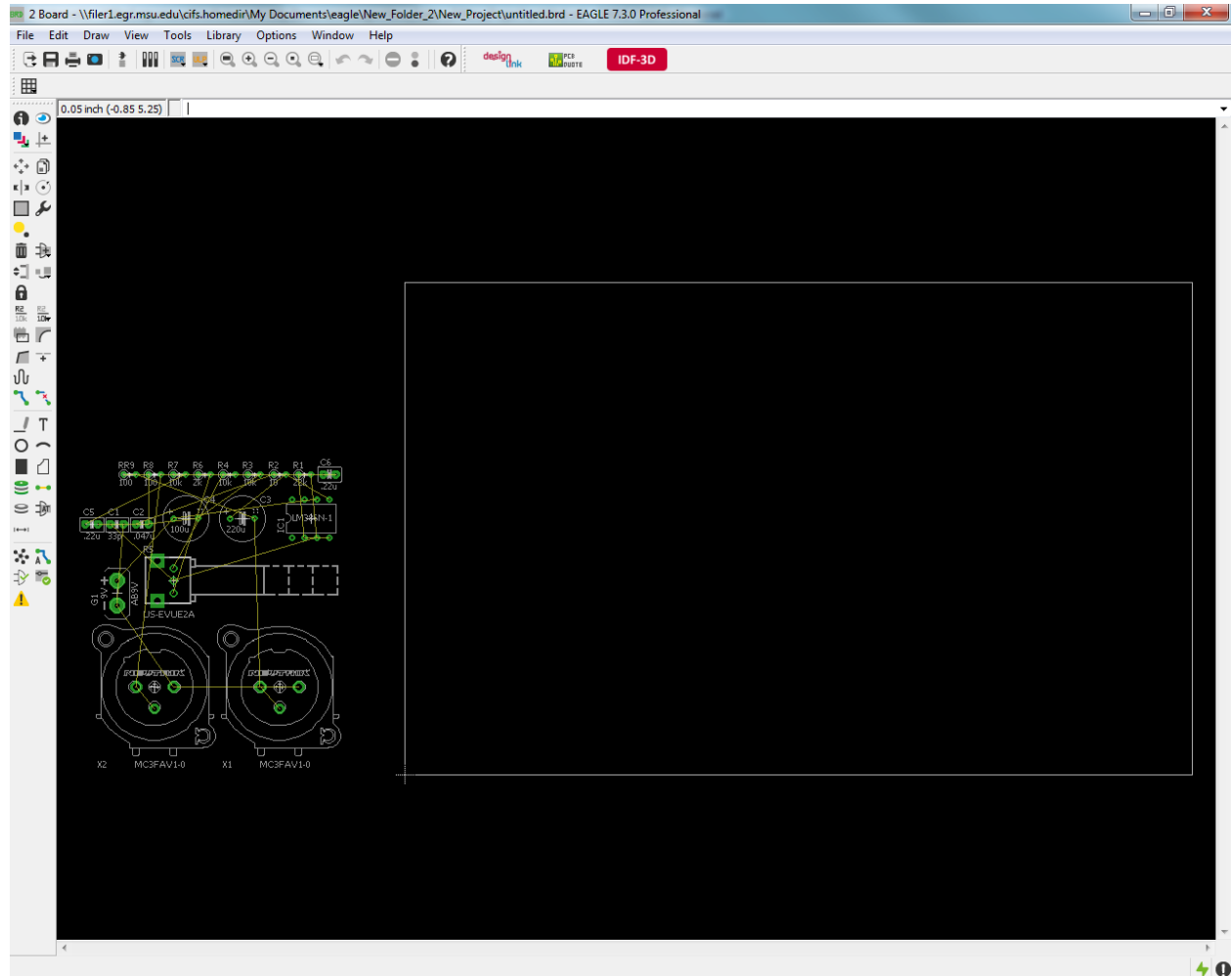


Figure 2

5. This is the screen that contains the board file. As shown in **figure 2**, the white outlines of the components are called footprints. These footprints show the outline of the components when they are mounted on the board. This allows the user to visualize the board layout and know where to place each component once the printed circuit board is fabricated. The green circles show the pads that the component will be connected to. Pads are where the pins of the component are soldered and the place where the traces connect to each component. The yellow lines show which pins need to be connected and provide a guide for routing the traces. These yellow lines do not physically connect pins but helps the user know where to route each trace. The large white box on the screen determines the size of the printed circuit board.

Laying out the board

1. To resize the printed circuit board it is possible to use the wire tool to create a box of the desired size. To delete the current box click the **delete** tool as shown in **figure 3**. Once the **delete** tool is selected click on all four of the sides of the box to delete it.



Figure 3

2. To create a box, it is necessary to check the units of measurement. Click the grid in the top left of the window as shown in **figure 4**. And set both of the drop down menus to inches.

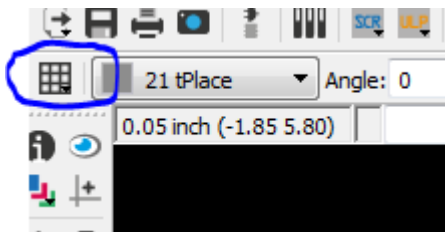


Figure 4

3. To create a new box select the **wire** tool as shown in **figure 5**.



Figure 5

4. Make sure the width is set to 0 before drawing any lines. This can be done with the dropdown menu at the top of the page.
5. Click and drag using the **wire** tool to create a box of the desired size
6. The next step is to move all of the components into the box and lay them out in a logical manner. It is usually beneficial to minimize the crossing of the yellow lines. This will help reduce the amount of traces that cross.
7. To move the components select the move tool as shown in **figure 6**.



Figure 6

8. Click and drag the components into the box.
9. The rotate tool shown in **figure 7** can be used to reorient the footprints

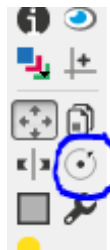


Figure 7

10. Once the components are laid out as shown in the example in **figure 8**, the traces can now be routed to form the electrical connections between all of the pads.

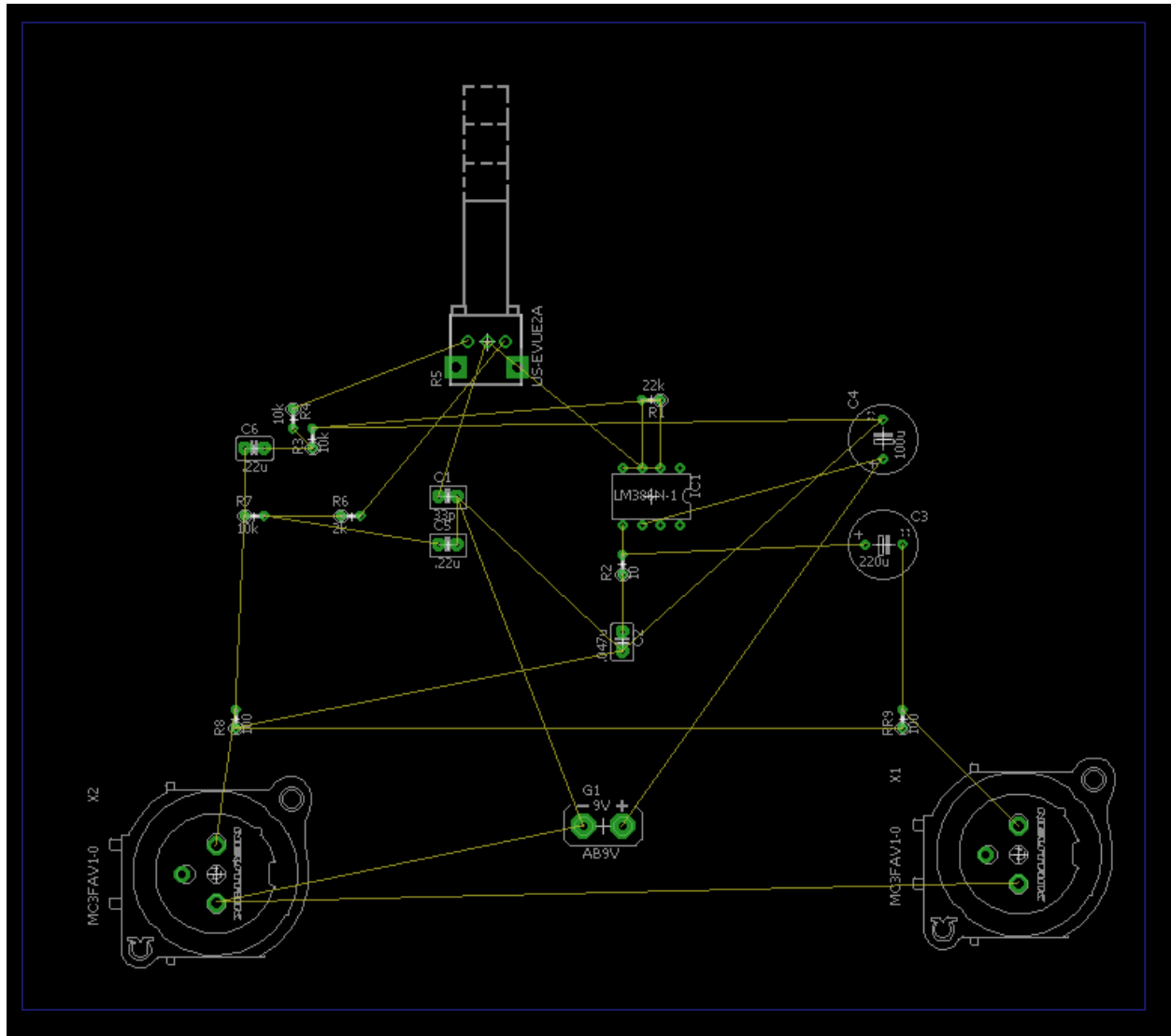


Figure 8

Routing the traces manually

1. Before any trace is able to be routed it is necessary to determine the width of the trace that is needed. The trace width is dependent on the material used, current through the trace, and thickness of the trace. For printed circuit boards fabricated in the Michigan State University ECE shop **table 2** [2] may be used to determine the trace thickness. This data uses the fact that the boards use 1 ounce copper for the traces. If the printed circuit board is being fabricated at another facility, the calculator found at, <http://www.4pcb.com/trace-width-calculator.html> [3], may be used to calculate the trace width for given parameters and board thicknesses. Calculating the correct thickness is essential to ensure that the traces don't burn up from overcurrent.

Temp Rise	10°C	20°C	30°C
Trace Width in Mil	Maximum Current in Amps		
10	1.0	1.2	1.5
15	1.2	1.3	1.6
20	1.3	1.7	12.4
25	1.7	2.2	2.8
30	1.9	2.5	3.2
50	2.6	3.6	4.4
75	3.5	4.5	6.0
100	4.2	6.0	7.5
200	7.0	10.0	13.0
250	8.3	12.3	15.0

Table 2

2. For the example circuit the trace width is 16 mil because the current in the circuit is never going to exceed 1.2A of current. To start routing a trace click the **trace** icon as shown in **figure 9**.



Figure 9

3. Once the **trace** icon is selected, the width can be changed to the desired value using the width dropdown menu as shown in **figure 10**. It is important to note the units that are being used.

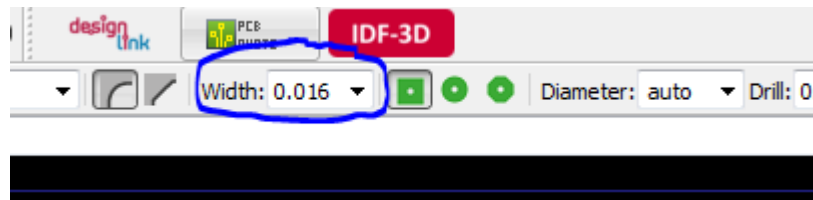


Figure 10

- At this point everything is ready to start connecting pads with traces. To start a trace, click the center of one of the green circles. A blue line should appear on the screen. Drag this blue line to one of the other pads that is connect to it with a yellow line. **Figure 11** shows a connection from R8 to R9 with a trace. This same basic process can be used to connect the rest of the pads in the circuit.

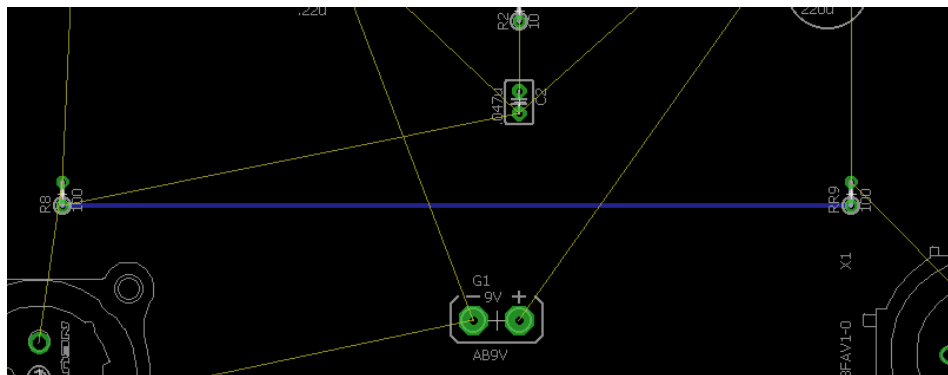


Figure 11

5. While connecting the other pads it may be necessary to change bend of the trace. This allows for more flexibility when routing traces. The toolbar at the top of the screen allows the user to choose from a variety of trace bends.
6. As the board becomes more crowded it may be impossible to simply just route traces around each other to get from pad to pad. Intersections of traces will occur as shown in **figure 12**. This common problem may be resolved by using a vias. Vias are when traces go from one layer of the board to another layer. There are a variety of boards that can be fabricated from manufacturers, but the ECE shop only has 2 layer boards. The boards

fabricated in the ECE shop also do not have plated vias which means that there is no electrical connection from one layer to another. There needs to be a wire soldered through the via hole to connect these two layers.

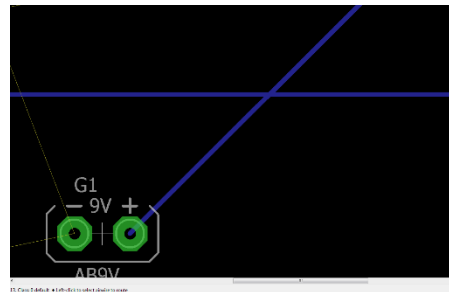


Figure 12

7. To create a via, start by running a normal trace, when the traces are about to intersect, left click the mouse to place the trace and then select the desired layer from the dropdown layer in the upper left corner of the screen. Eagle will automatically place a via and the trace will be on the desired layer. **Figure 13** shows a trace that starts on the bottom layer and then crosses over to the top layer of the board. The via is denoted by a green square with a circle in the center of it.

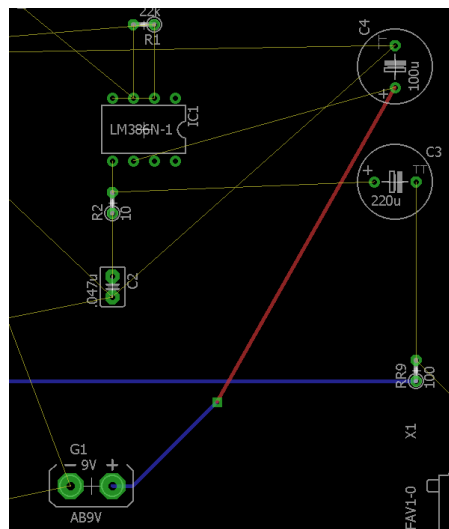


Figure 13

8. When routing traces it is perfectly fine to overlap two traces on separate layers, but overlapping a trace with an undesired pad is unacceptable. Because these green pads are through hole, this pad will include a hole that has access to all of the layers of the board. Holding down the alt key while routing a trace will enable the alternate grid, allowing for finer control of the trace location and will help avoid this mistake.

Routing traces with the Autoroute tool

1. It is advised that circuit boards are routed manually if possible, but a helpful tool for creating printed circuit boards quickly is the autoroute tool. The autoroute tool will automatically route the traces on the printed circuit board and can be used in addition to manually routing traces. Before using the autoroute tool a "Design Rules" and "Autorouter Parameter" file may be uploaded to Eagle to guide the automated process.
2. To use the design rules and autorouter parameters for fabrication in the Michigan State ECE shop, navigate to <https://www.egr.msu.edu/eceshop/pcb/programs.php> and download the zip file at the bottom of the page.
3. Extract the files in this zip file to a convenient place.
4. Open the board editor and click on the **DRC** icon as shown in **figure 14**.



Figure 14

5. Click the **Load** button and select the "MWSEagleDesignRules.dru" file and then close the design rules box.
6. To load the autorouter parameters file, start by clicking the **Auto** icon as shown in **figure 15**.



Figure 15

7. Click the **Load** button and select the "MWSEagleAutorouterParams.ctl" file and close the autorouter box.
8. At this point the program has all of the information to finish routing the traces without causing any major problems.

9. To start the autorouting process, click the **Auto** icon and click “ok”. Eagle will give multiple options of layouts to choose from. **Figure 16** shows a possible routed layout. The autorouter tool is a powerful tool that offers much customization. For more information go to <http://www.build-electronic-circuits.com/eagle-autorouter/> to learn how to use this tool to its fullest extent.

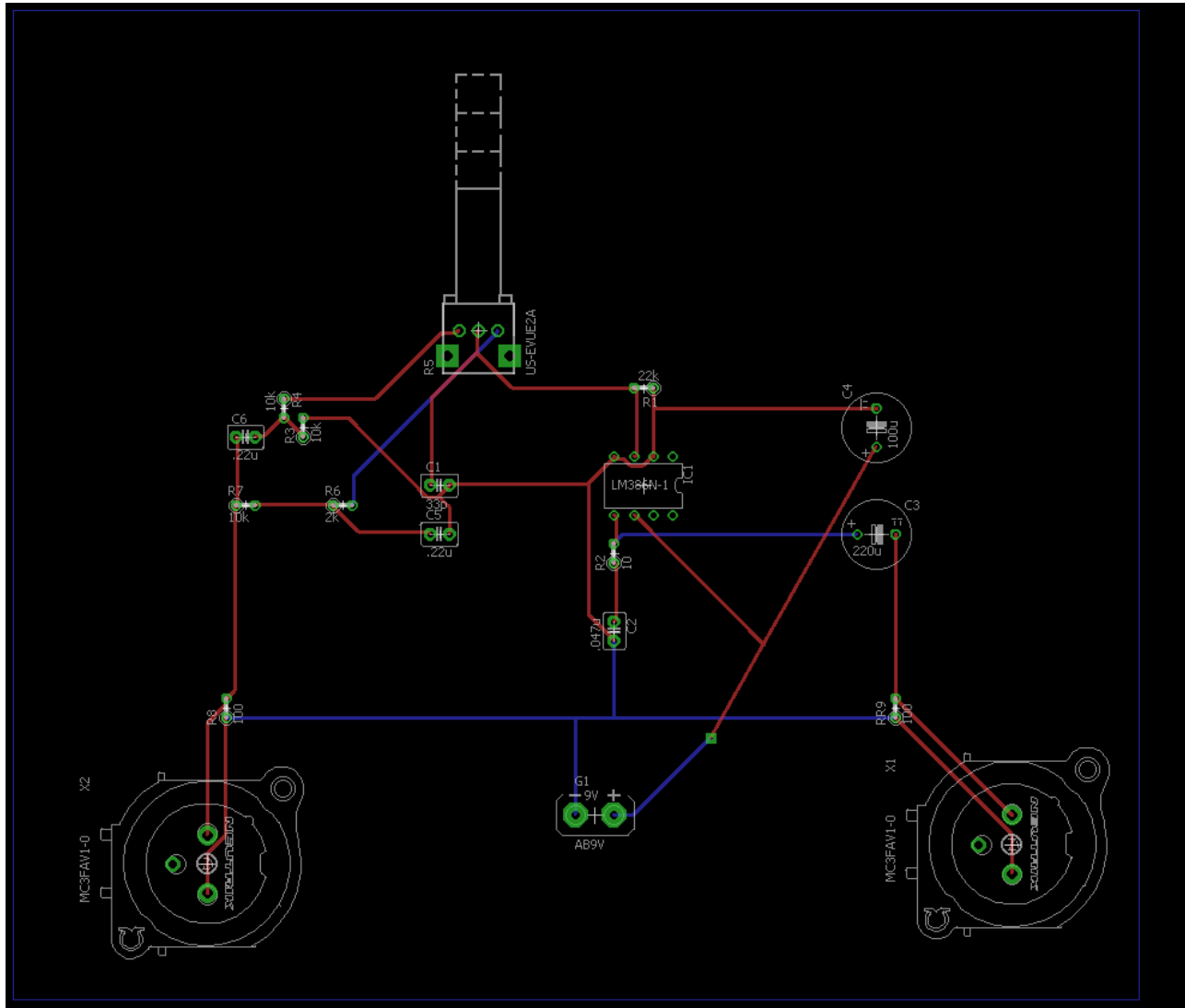


Figure 16

10. At this point it is a wise idea to check the board for errors. Click the **DRC** icon and resolve any necessary errors.

Finalizing the board

1. Once the layout and traces are finalized it is time to add copper pours. In this example, the copper pour is used to create a ground plane. This allows for a concrete ground reference for the entire board.
2. Start by click the **Polygon** tool on the left side of the screen
3. Next, switch to the top layer by using the dropdown menu in the top left of the screen.
4. The **isolate** setting is used to give a clearance between the pour and the other signals on the board. For this example .012" is used as shown in **Figure 17**. Make sure the width is set to .01. With the **Polygon** tool selected, draw a box right on top of the dimension box that was already created. A box with dotted lines should appear as shown in **figure 18**.



Figure 17

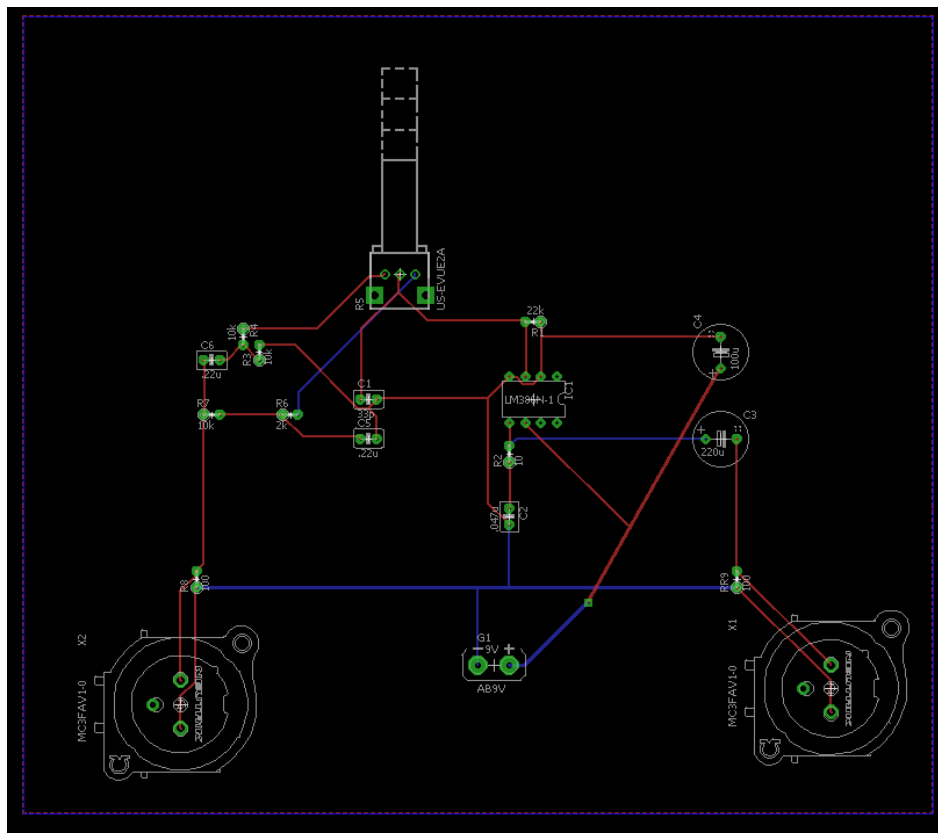


Figure 18

5. The next step is to name this polygon by using the **name** tool from the toolbar. With the **name** tool selected, click on the box that was just created and name it "GND". This will allow the program to know that this plane is the GND net that was created in the schematic.
6. Once the polygon is named, click the **ratsnest** tool to fill in the polygon. **Figure 19** shows the **ratsnest** tool and **figure 20** shows the final printed circuit board.

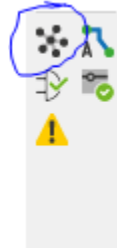


Figure 19

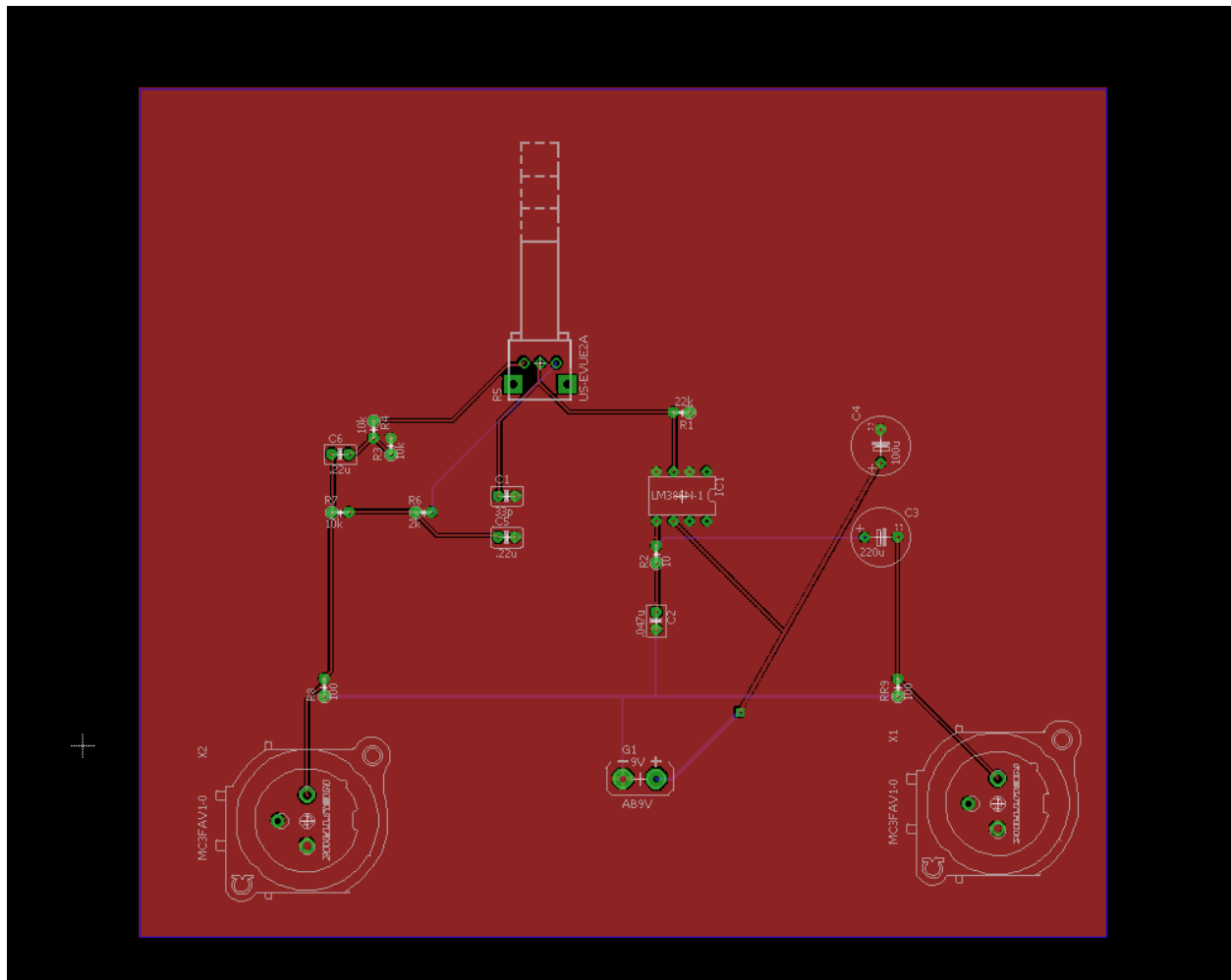


Figure 20

Exporting the board for production

1. Once the design is finalized, gerber files need to be produced. Gerber files are universal files for printed circuit boards that most fabricators use.
2. To have a printed circuit board fabricated in the Michigan State ECE shop the following files are needed. Top layer (component side) ("*.CMP") , Bottom Layer (solder side) ("*.SOL"), Board outline ("*.BOL"), and Drill file ("*.DRD").
3. To create these files navigate to <https://www.egr.msu.edu/eceshop/pcb/programs.php> and download the Eagle CAM job File.
4. Unzip the file into the director "Eagle/cam" if possible. If not just unzip it to a convenient location.
5. Open the board editor and click **File, CAM Processor, File, Open, Job**, and select "gerb274xECEShop.cam"
6. Click the **Process Job** button and the files should now be in the same directory as the board files.
7. This board now completed and the necessary files for fabrication are produced.

Conclusion

Printed circuit boards are a necessary part of most hardware projects. They allow for a robust final design that is easily mass produced. The board layout feature in Eagle allows for the creation of a variety of printed circuit boards for different applications. This tutorial uses the “simple bass boost headphone amp” circuit to demonstrate the general process of using a schematic file to create a board file in Eagle. To obtain more information on creating a schematic file navigate to <https://learn.sparkfun.com/tutorials/using-eagle-schematic> for an in depth tutorial.

References

- [1] "Make" Simple Bass Bump Headphone Amp project: <http://makezine.com/projects/bass-bump-headphone-amp/>
- [2] ECE shop website: <https://www.egr.msu.edu/eceshop/>
- [3] Trace width calculator: <http://www.4pcb.com/trace-width-calculator.html>
- [4] Autorouter information: <http://www.build-electronic-circuits.com/eagle-autorouter/>
- [5] Schematic file tutorial: <https://learn.sparkfun.com/tutorials/using-eagle-schematic>