

IEEE SP17 EAGLE Workshop Handout

EAGLE is a powerful CAD software used to design circuit schematics and lay out PCB boards. In this workshop we will go over basic schematic creation and basic layout principles, and use those principles to create a board that serves as a 12V-to-9V DC-DC power converter.

The procedure for creating a custom part and project details can be found in the Appendix section at the end of this handout.

Let's begin!

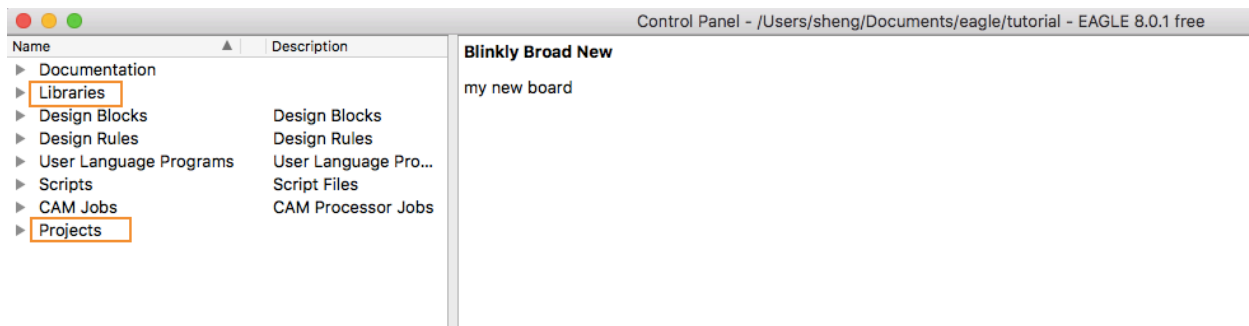
1. The first step is to install and set up EAGLE.

- a. Download EAGLE at: <http://www.autodesk.com/products/eagle/free-download>
- b. Install EAGLE and create an Autodesk account.
- c. The IEEE Github page has the library file we need for our project: go to <https://github.com/ieee-uiuc/eagle-workshop-sp17> to download **ieee.lbr**
 - i. Feel free to try to recreate the custom part we supply for you in your own time; we are skipping this step due to workshop time constraints.

For the purpose of this workshop, let's start with creating a custom component in a new library. You'll most likely do this often in the future, as the components that you may need are not always included in EAGLE's default libraries. Please see Appendix A at the back of this handout for instructions on how to do this.

2. The second step is to create a new project and import the library files.

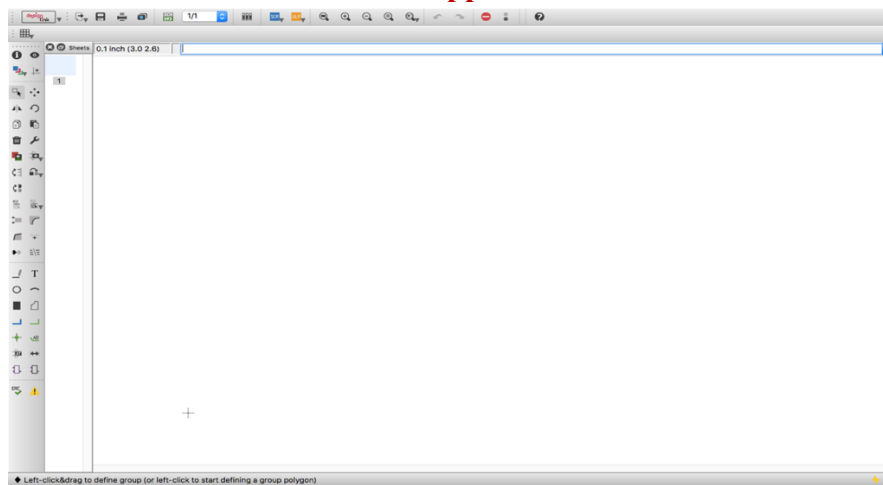
- a. Open EAGLE; this will bring you to the EAGLE's Control Panel. Go to **File>New>Project**; then a "New Project" will appear under the **Project>eagle** section. Rename the project with an appropriate name (ex. "IEEE EAGLE Workshop SP17").
- b. Edit the description of the project by right clicking the newly created project and choose **Edit Description**; the descriptions can be entered in HTML format. For example: **EAGLE Workshop Power Board**.
- c. Before we start connecting our circuit, we first need to import the customized component libraries. Put the "ieee.lbr" file you previously downloaded into the lbr folder of your EAGLE installation path <Applications/EAGLE-8.0.1/lbr>. In the Control Panel, expand the **Libraries** tab and see if the library file, called **ieee.lbr**, exists in the tree view (close and reopen the "Libraries" tab if it does not show up). If it does, right click **Libraries** and select **Use All**. The libraries can now be used in our project; we have now finished setting up EAGLE.



The EAGLE Control Panel

3. The third step is to draw the schematic for our board. This defines how the components on the board are connected. (Tool Bar layout can be found at the end of this section. You can also hover your mouse over icons in the Tool Bar to see their names.)

- a. Create a new schematic of the project by right clicking the project, select **New>Schematic**; save the schematic by **File>Save**. The window below should show up after we create a new schematic. **Find the Schematic in the Appendix of this handout.**

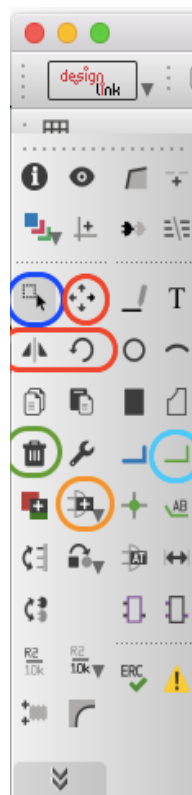


Schematic Window

- b. On the tool bar (shown below), click **Add**. This brings up a window. Look for the **ieee.lbr** library, select **IR1209SA**, double click, then click anywhere inside the schematic workspace to place it.
- c. After placing the component, hit the **“Esc”** key once on your keyboard to return to the **Add** window; repeat the last step to add **three MTA-100** and **two C_10μF**. Finally, search for the **fuse** library and find the part called LITTLEFUSE. Add two of them onto the schematic, then hit **“Esc”** twice to exit the **“Add Component”** mode.
- d. Now we want to connect the components together using wire. Select **Net** in the menu tool bar and connect the components according to the schematic in Appendix B. Use **Move, Mirror or Rotate** to adjust the orientation of the components: make sure the orientation is exactly as the schematic shows, and *that the right ports numbers are connected together*. You can also use the **Junction** tool (the green dot) to explicitly specify a connection; while our board is simple, it

becomes necessary for complex circuits with a lot of crisscrossing wires. While using **Net**, you can also click on white space to route it exactly as you want it to.

- e. Finally, we need to specify the names and values of our components (the fuses and the capacitors). To do this we right click the components and go to *Attributes* or *Properties*, giving the components relevant names (like C1 & C2 for the capacitors) and specify their values; doing so helps us find the components in the Board View much faster. Alternatively, you can also use the **Name** and **Value** tools in the Tool Bar; click on the appropriate component to rename them.
- f. We can additionally check for problems with your circuit using the *ERC* tool (Electrical Rule Check). Click it and see if any errors pop up. Feel free to ignore any irrelevant ones and deal with the important ones.



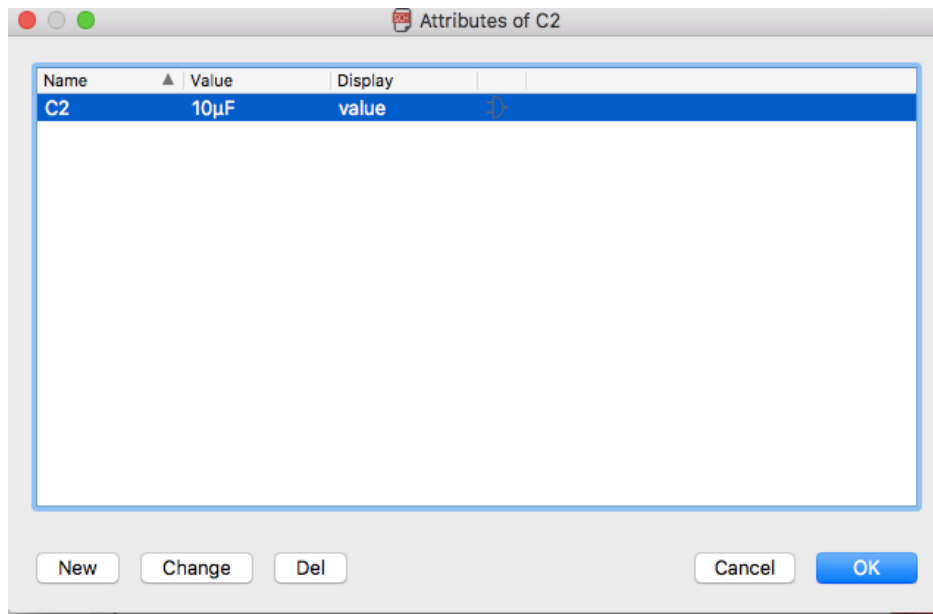
Tool Bar, from left to right, top to bottom:

Select, Move

Mirror, Rotate

Delete, Net (Wire)

Add



Attribute Menu of Capacitor C2

4. The fourth step is to design the layout of our PCB board.

(Again, Tool Bar layout can be found at the end of this section)

- a. Generate a PCB board use **Generate/switch to Board** button on the menu bar. If the program asks to create it from the schematic, hit OK.

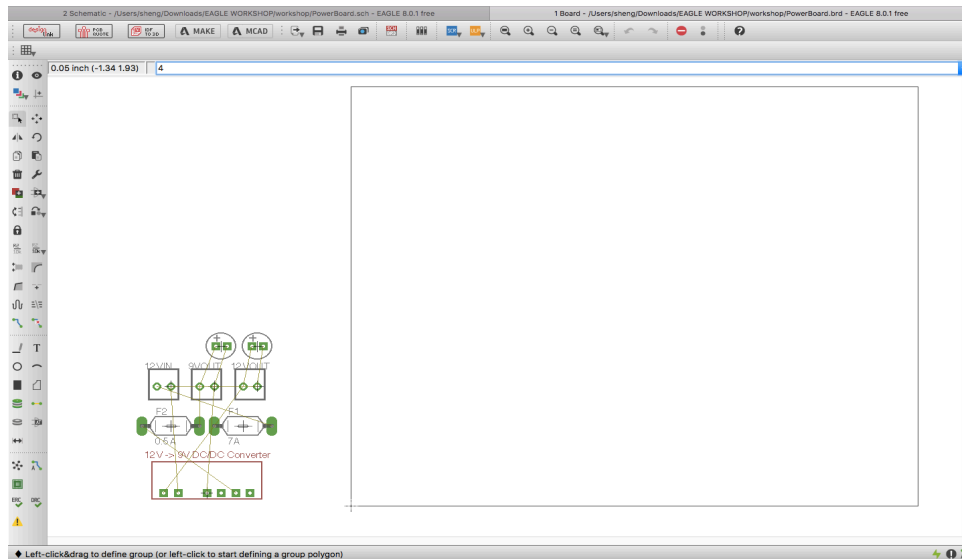


Generate board file from schematic

- b. EAGLE opens a new window, and generates a *.brd file in our project folder. Use **Move Group** to move the **entire schematic into the board area (the big square shown below)** by first selecting the entire group of components using **Group and** with **ctrl + right click** to drag the entire group into the board area. You can also do the same thing by highlighting all the components and dragging the entire group into the board area.

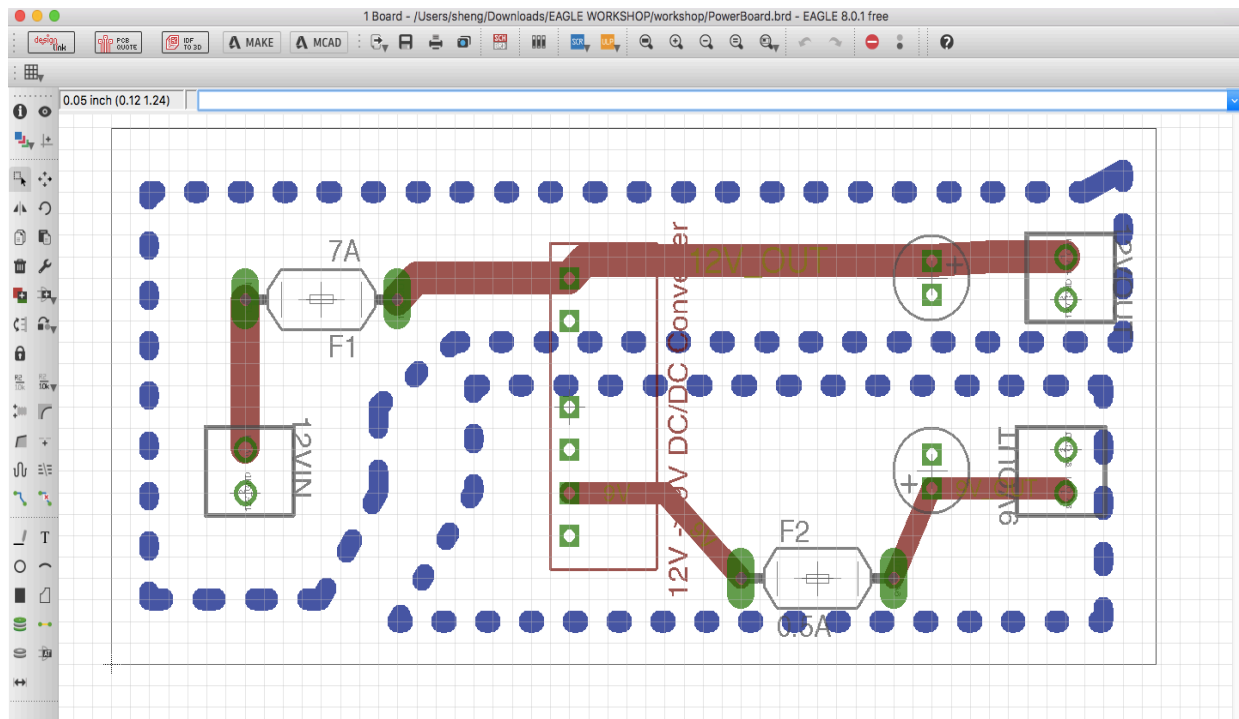
Alternatively, you can also drag one component in at a time, and place it according to your preference for optimal layout.

Additionally, use the **Move** tool to resize the board by dragging the boundaries to an appropriate size.



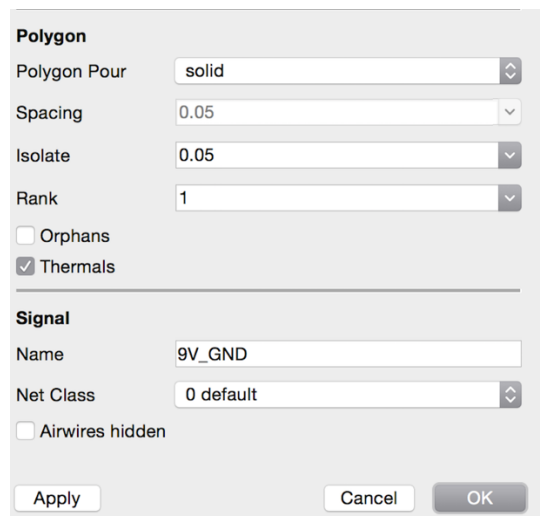
EAGLE Board Window

- c. The yellow lines indicate which ports need to be connected according to your schematic; use that to dictate your board layout. Once a connection is made, the appropriate yellow line disappears. To connect the components, use the **Route** tool; to delete routes you want to change, use the neighboring **Ripup** tool. Finally, remember to select the correct layer that you are routing on (i.e. top or bottom, at the drop-down menu next to **Grid**) – this is indicated by the color of your route.
 - i. Throughout routing, configure layout of the components using the same techniques that we used in creating our schematic, i.e. **Move**, **Mirror** and **Rotate**. You can also hit **RatsNest** throughout placement; EAGLE will rearrange the yellow lines to their shortest length for a clearer view (**this is optional and will not wire the board**, it is just for a clearer view of the connection relationship).
 - ii. The traces widths need to be considered: wider traces are necessary for power lines. See Appendix B for the trace width.
 - iii. If you notice that a yellow connection is wrong, then you need to fix it in the schematic; any changes in the schematic will be immediately reflected in the board.
 - iv. Try to complete this part by yourself; it's just completing a puzzle! The goal is to minimize the area. If you're lost, see Appendix B for a completed trace layout.
- d. Next, we want to fill a certain area with a layer of copper to create a ground plane (we could also do this for VCC, but we don't need to for this project). To do this, use the **Polygon** tool to draw lines and create a closed polygon. We want to do this for the ground layer, i.e. the bottom: thus, **make sure the bottom layer (blue) is selected**. In this project we fill two areas with the **Polygon** tool as shown in the Figure below. When finished, hit **RatsNest** to fill the selected area entirely with copper; the area selected by the polygon will be colored in the corresponding layer color (i.e. blue for the bottom layer).

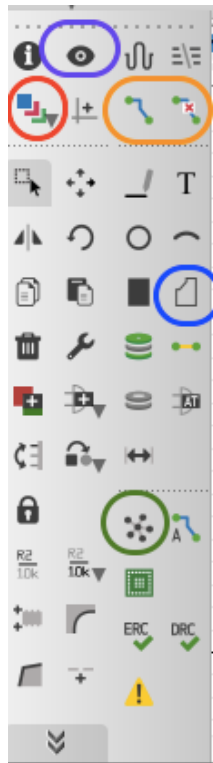


Outline of the two blue polygons

- e. The copper layer we just created (on the bottom layer) is often used as a common connection area for multiple components' pins, such as VCC or GND, in this project, we will use the two layers as 12V_GND and 9V_GND respectively. **Right Click** the polygon boundaries and go to **Attributes**, and name the polygons. **Details on names for each component can be found in Appendix B.**
 - i. Additionally, you need to change the **Isolate** parameter in **Properties**. This is the parameter that determines how much separation exists between the fill and other signals on that level. If you make it too small, you risk coupling signals together. A decent value is 50mil (= 0.05 inch).

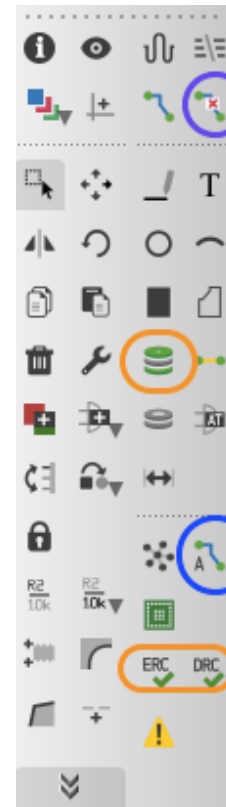


Polygon Attribute Menu



*Tool Bar for Board View.
From left to right, top to bottom:*

View
Show Layers
Net, Ripup
Polygon
RatsNest



*Additional tools worth mentioning.
From left to right, top to bottom:*

RipUp (Delete traces)
Via
AutoRouting
ERC, DRC

- f. Finally, we need to check if we've laid out anything strangely, or if we did anything that isn't possible to be manufactured (i.e. you'll be hard-pressed to find a manufacturer that can do nanometer spacing). You will usually download a manufacturer's DRC rules, and click the **DRC** tool to check that your layout conforms to their specifications. We don't supply a DRC file for this workshop, but be sure to complete this step when designing your own board!

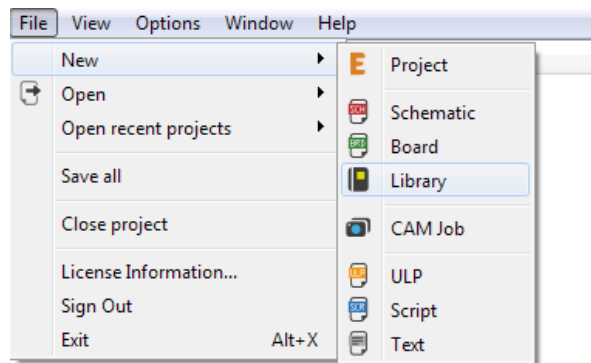
You've successfully completed our workshop! Here we explain a couple of other tools:

1. If we want to connect traces on different layers together, we can use a **Via** that goes through layers.
2. The **Auto Router** automatically routes up the traces for us, but the solution it produces is usually far from optimization, so we tend to not use it.

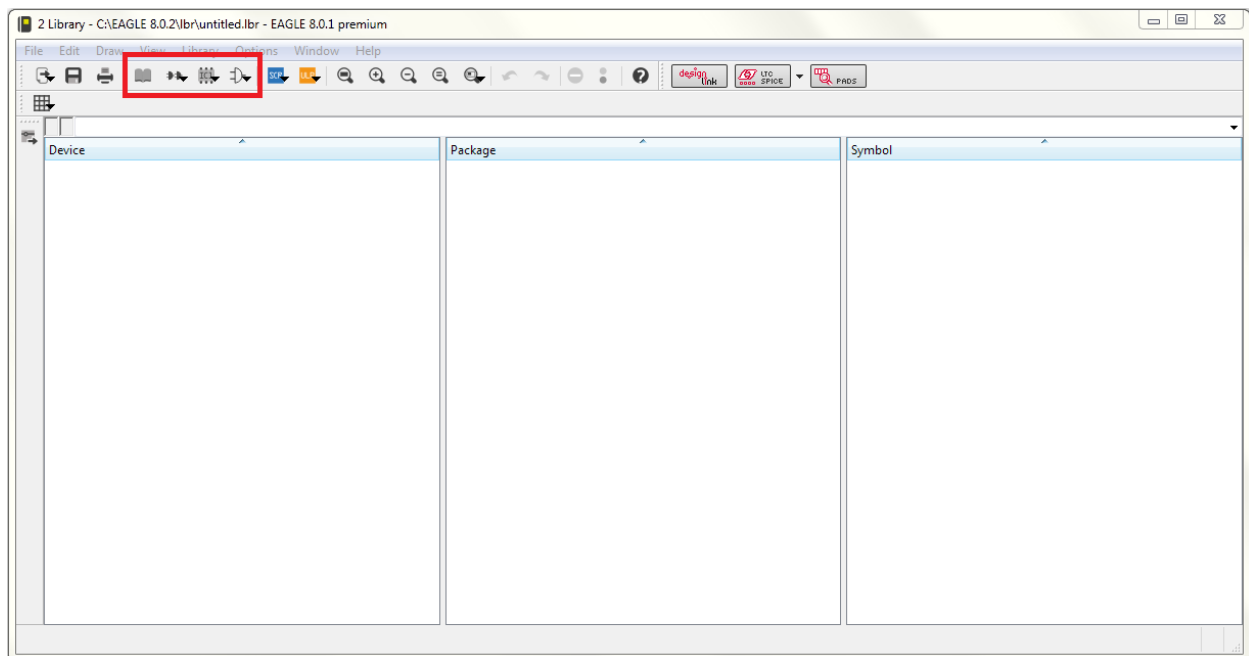
Appendix A: Creating Custom Parts

Although EAGLE has many available libraries, you'll occasionally find that a part that fits perfectly in your project's specs does not have a corresponding EAGLE library; in that case, you have to make a custom part. This appendix section will guide you through the process of creating a simple custom part for use in the main project, as well as to serve as a reference for any of your future projects.

1. In the EAGLE Control Panel (i.e. the main window), go to the upper-left corner and go to **File>New>Library**, as shown below.



This opens up the Library Editor window, shown below:

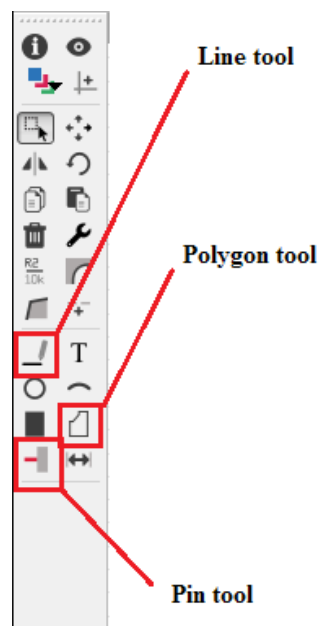


The four main buttons (boxed in red) are (from left to right): **Table of contents**, which we'll go back to after each sub-part's creation, **Edit Device**, **Edit Package**, and **Edit Symbol**. We'll use these three in turn to make a complete part.

2. Before we actually create anything, look at the datasheet PDF that was given to you in the IEEE Github page. This is the datasheet for the part we are creating (Note that the part number is the document title).

It's important that you are comfortable with looking at datasheets, as you'll have to look at lots of them to choose the parts for the circuits you design. Therefore, this guide will be purposefully vague in providing details that can be obtained from the part's datasheet.

3. Once you've looked at the data sheet and found the necessary part dimensions and connection info, you can begin creating the part. First we'll create the part's symbol: click on **Edit Symbol**. A dialog box will open, prompting you to either choose an existing symbol to edit or to create a new one. We want to create a new symbol, so enter the model number of the part into the box and click **OK**.
4. The library editor window will turn into a schematic editor. Along the left side of the window, there is a toolbar which has all the necessary things you'll need to create this schematic.



To create the schematic, use the **Pin** tool to add all the necessary connections. Then, draw a box (using the line tool or the polygon tool) big enough to fit all the part's connections. **Make sure to name all the connections!** Once you're done, press **Ctrl+S** to save, then click on the **Table of contents** button to return to the main window.

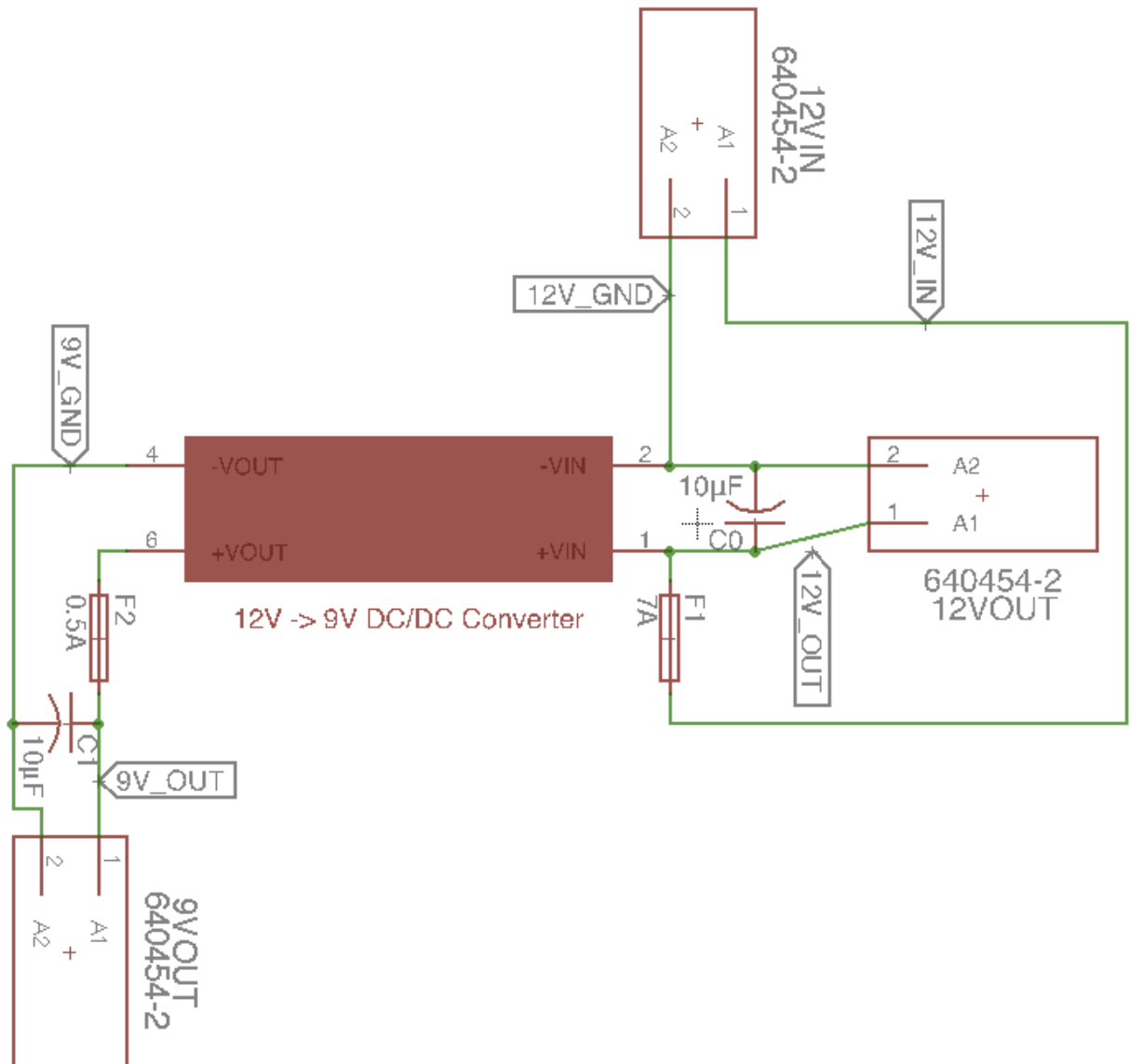
5. Now repeat this process to create the package (which is the footprint that is used during the PCB layout). Click on the **Edit Package** button; a package editor window will appear. It is a very similar environment to the schematic editor, with the addition of two new buttons in the bottom left of the toolbar: the **Pad** tool and the **Hole** tool. **Pad** is used for creating surface-mount, and **Hole** is used to create through-hole connections for the part. Once you're finished drawing the part *to the right dimensions from the datasheet*, use the correct tool to add the right type of connections to the part (surface-mount or through-hole). Once you're done, save and return to the main window.

6. Now that you've created both the schematic and the package, it's time to create the device that these will be attributed to. Click on the **Edit Device** button, and create an appropriately-named device. This will turn the window in to a device editor, where the left side of the window will manage the symbol, and the right will manage the package. To add a symbol, click the **Add** button in the left-side toolbar and choose the symbol you just made (it should be the only one in the list that pops up). To add a package, click on the **New** button near the bottom-right corner of the window and choose the package you just made.
7. To complete the device, we need to associate the connections in the schematic to the correct connections in the package. We use the **Connect** button near the bottom-left corner. This brings up a window that lists all the connections from both the symbol and the package. Select two corresponding connections (one from each list), then click the **Connect** button to connect them. Do that for each relevant connection (there may be some unused pins, for example), then save when you're done.
8. Congratulations! You've created a library!

Appendix B: Project Details

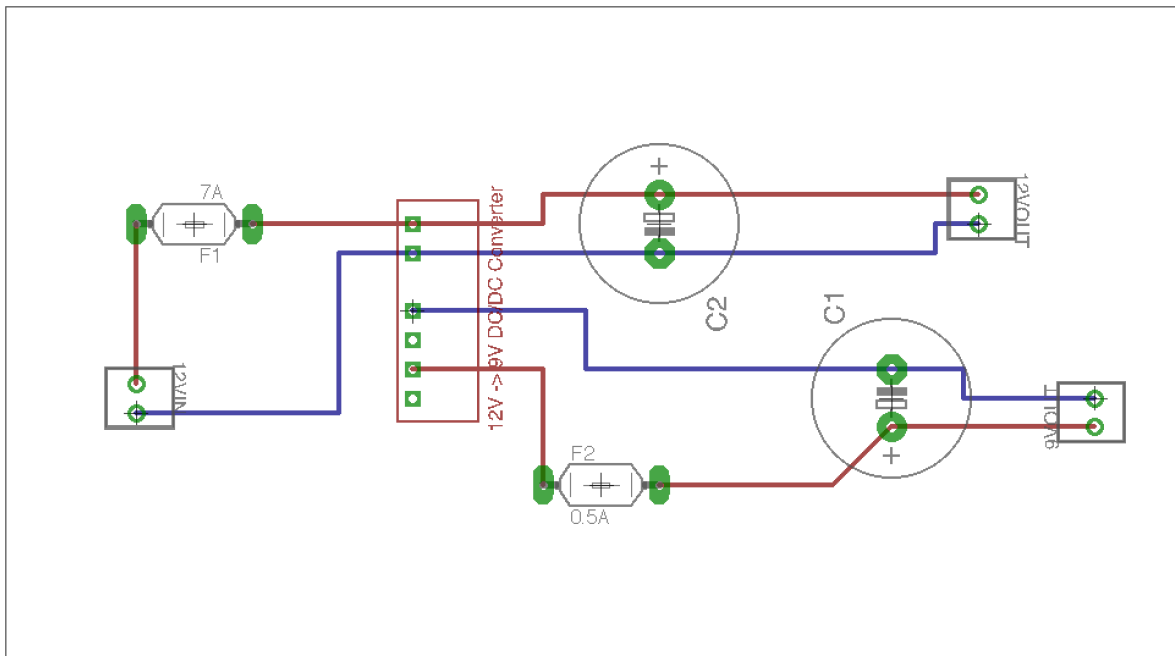
Schematic View

With wire labels, component names and values:

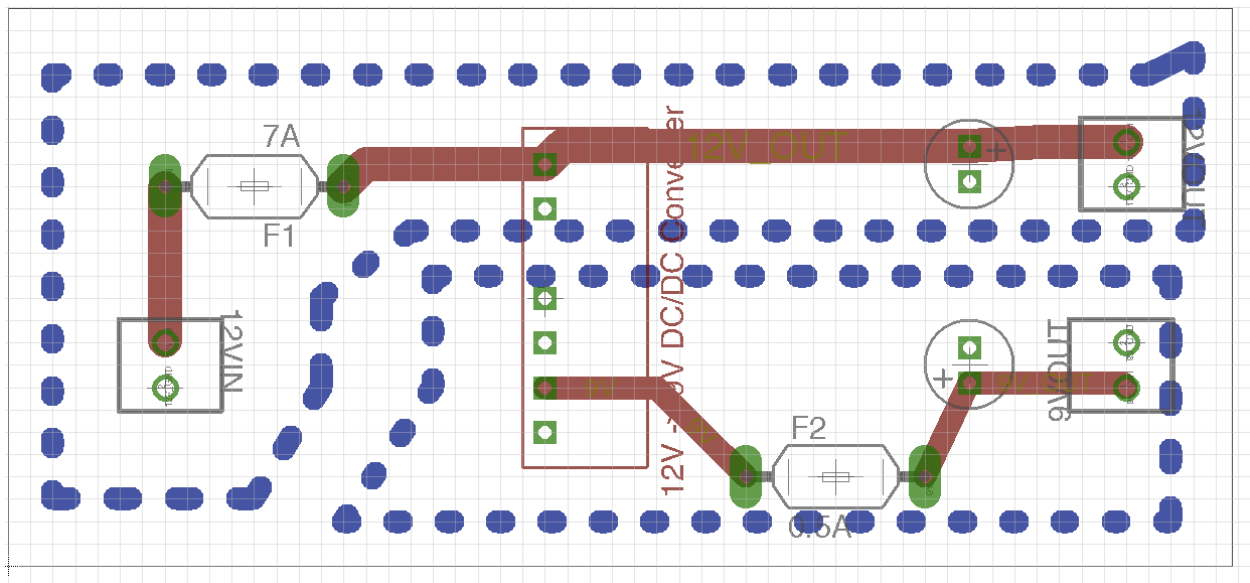


Board View

Traces:

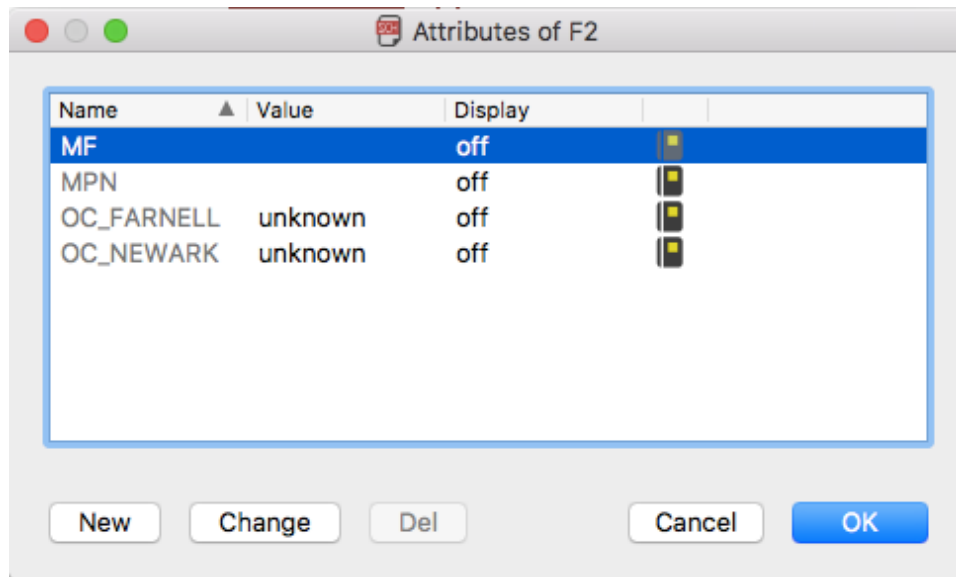


Top layer is in red and bottom layer is in blue. We can see that the polygon is drawn onto the bottom layer:



Attribute Examples

Fuse:

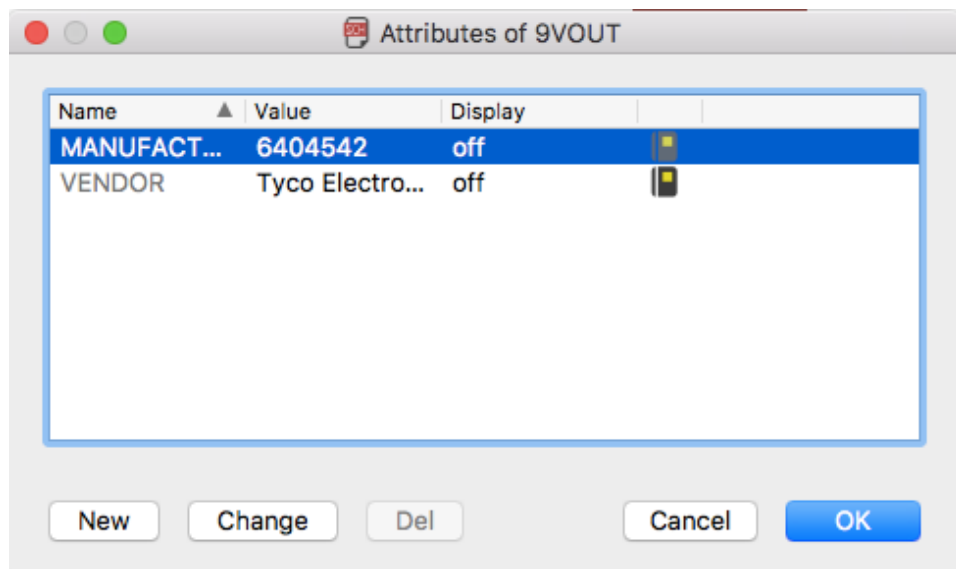


A dialog box titled "Attributes of F2" with a table of attributes. The table has columns for Name, Value, and Display. The first row is highlighted in blue. Below the table are buttons for New, Change, Del, Cancel, and OK.

Name	Value	Display
MF		off
MPN		off
OC_FARNELL	unknown	off
OC_NEWARK	unknown	off

New Change Del Cancel OK

MTA-100:



A dialog box titled "Attributes of 9VOUT" with a table of attributes. The table has columns for Name, Value, and Display. The first row is highlighted in blue. Below the table are buttons for New, Change, Del, Cancel, and OK.

Name	Value	Display
MANUFACT...	6404542	off
VENDOR	Tyco Electro...	off

New Change Del Cancel OK

Lower Polygon Properties:

Wire

From 3.95 1.35

To 1.2 1.35

Length 2.75

Angle 180

Width 0.05 ▾

Cap round ▾

Layer 16 Bottom ▾

Curve 0

Polygon

Polygon Pour solid ▾

Spacing 0.05 ▾

Isolate 0.05 ▾

Rank 1 ▾

☐ Orphans

☒ Thermals

Signal

Name 9V_GND

Net Class 0 default ▾

☐ Airwires hidden

Apply Cancel OK

Upper Polygon Properties:

Wire

From 0.35 2.8

To 3.7 2.8

Length 3.35

Angle 0

Width 0.05 ▾

Cap round ▾

Layer 16 Bottom ▾

Curve 0

Polygon

Polygon Pour solid ▾

Spacing 0.05 ▾

Isolate 0.05 ▾

Rank 1 ▾

☐ Orphans

☒ Thermals

Signal

Name 12V_GND

Net Class 0 default ▾

☐ Airwires hidden

Apply Cancel OK