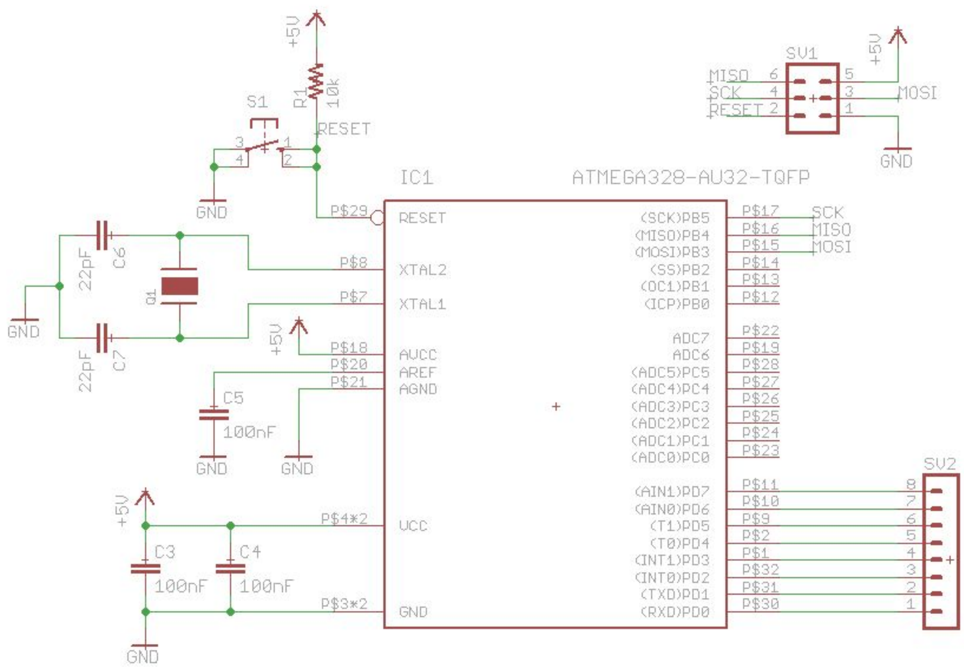
Michigan Embedded Systems Hub

LESSON: Introduction to Eagle
Part 2: Layout Design

Colin Szechy, Jonathan Kurzer, and Sam DeBruin

December 20, 2015



1. Introduction - see lecture slides!

Topics covered in the lecture:

- Board outlines and dimensions
- Placing components
- Routing traces
- Exporting Gerber files

The lecture slides can be found here.

Check out Eagle Part 1 for schematic design!

2. Basic EAGLE Use

If you are new to EAGLE, there are a few things that you should know before you get started:

- Movement within a schematic, board, or library is achieved by clicking and dragging with the middle mouse button. If your middle mouse button is unavailable or you just don't like to use the middle mouse button, type "set Interface.UseCtrlForPanning 1" into the command line. Then hold the Ctrl key to pan.
- Zooming in and out is achieved by scrolling the mouse wheel.
- Tools can be used either by pushing the button or typing the corresponding command in the command line.
- When using a command, return to the default by pushing the STOP button or by typing a semicolon followed by a carriage return.
- Nets named with the 'name' command in schematic will automatically be connected to nets with the same name elsewhere in the schematic.
- To select an object in a tight space filled with other objects, click near the object's origin. The object will highlight but not yet capture. Left click again to confirm or right click to cycle parts in the area. Left click anywhere in the schematic whenever the correct part is highlighted to capture it. This is more useful in the layout editor than in the schematic editor.
- When searching for a part in the 'add' dialog, the '*' symbol represents a wildcard. For example, the part 'MA03-2' will successfully be found by the search '*03-2' but not by the search '03-2'.
- If you are confused by any of the above items, ask your MESH instructor.

3. Initialize Board

When first initializing EAGLE, one will find themselves on the Control Panel screen.

First, we'll load the *part1.sch* file from the 'lesson 2':

- From the EAGLE Control Panel, select File → Open → Schematic
- With in the 'lesson 2' folder, open '*part1.sch*'
- After the schematic has loaded, type **'board'** or press the  button in the top-left-hand corner.

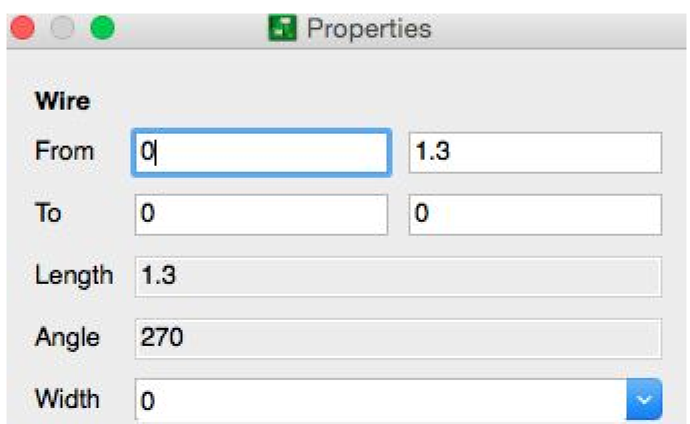
If this is the first time opening the board, the program will prompt you to create the board from schematic. Click yes.

4. Board Outline

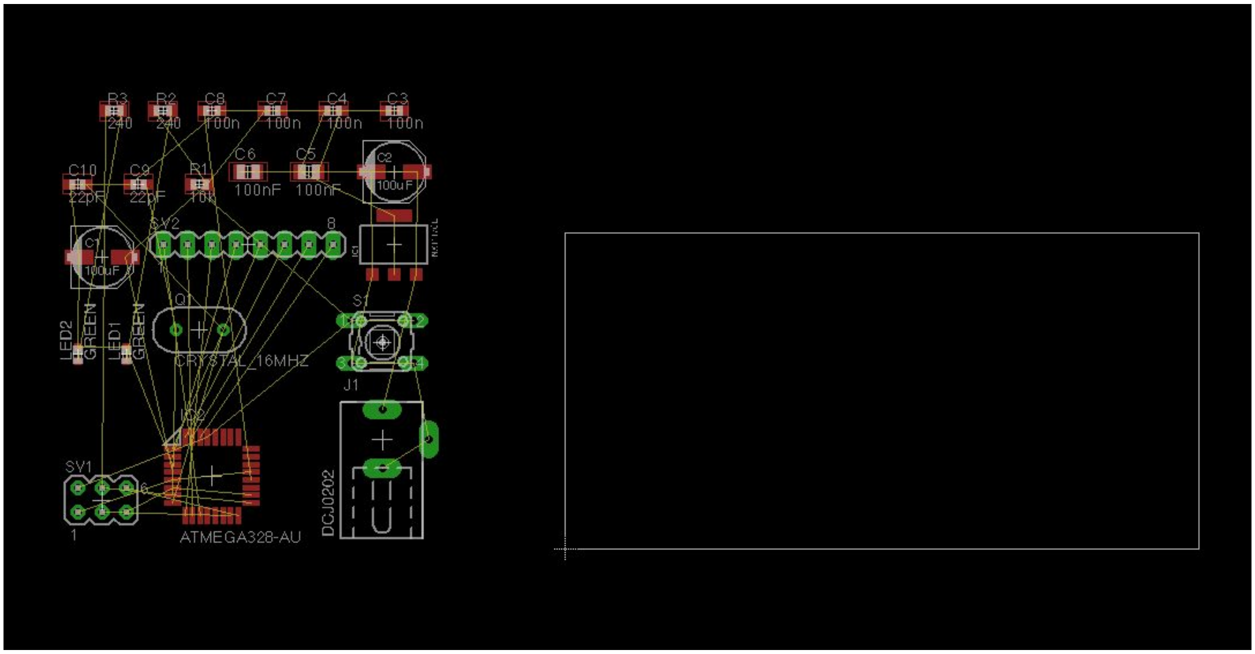
First, we will adjust the outlines of the board - which are in white, on layer 20 - to dimensions that will fit all of our components - specifically, a 2.6" x 1" rectangle with its bottom left corner at the origin.

(**NOTE**: On free versions of EAGLE (which are available for free for anyone to download, unlike the full professional versions of EAGLE on CAEN), all parts of the board must be placed within a 4" x 3.2" space with the bottom left corner at the origin; the program will refuse to place anything outside of that space.)

- Use the **'info'** command or press the  button (to open the properties menu), and select the left of the four sides of the board.
- Change the second 'From' entry to 1.3. Ensure the width of the wire is '0' (at the bottom of the screenshot to the left).
- Use the **'info'** command, select the top side, and change the first 'From' entry to 2.6, the second 'From' entry to 1.3, and leave the first 'To' entry at 0 and the second 'To' entry at 1.3. Ensure the width of the wire is '0'.



- Use the **'info'** command, select the top-right side, and change the first ‘From’ entry to 2.6, and leave the second ‘From’ entry to -0, the first ‘To’ entry at 1.3 and the second ‘To’ entry at 0. Ensure the width of the wire is ‘0’.
- The layout will look like the bottom image when you are finished.



5. Disperse Components

With the outline placed, the next step is to arrange the components within the board area, using the “ratsnest” view. The yellow lines are named “airwires”, and represent the connections that need to be made by routing the “nets”, or connections made in the schematic.

Next, we will be placing components to minimize airwires and simplify the routing job ahead of us. The process required for this step - placing components, making modifications, rearranging them - is essential for an efficient and practical layout of the board (before routing traces begins).

We will start with the regulator, IC1, and the microprocessor, IC2 and group these items separately and with enough space for support components.

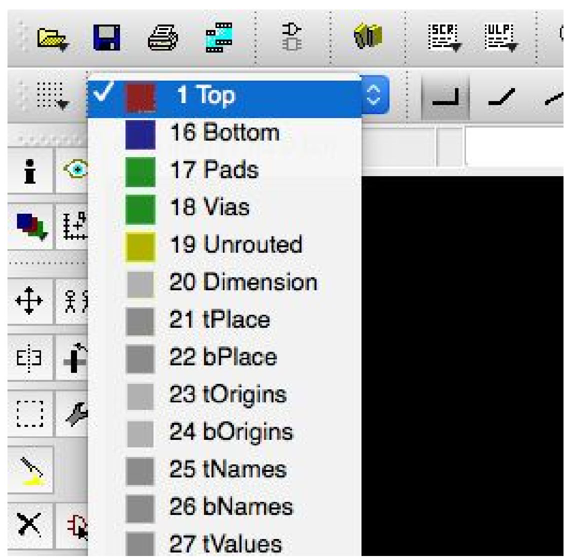
- Use the **‘move’** command or press the  icon. Then, click the light grey cross in the middle of IC1/ the regulator, which will turn white, and move the mouse to where you want to place the regular inside of the board dimensions and click again.

- Now, **'move'** IC2/the microprocessor - but place the regulator and the microprocessor separately, and with enough space for support components.
- **'Move'** the microprocessor's decoupling capacitors (C3, C4, C7, C8) as close to the microprocessor as possible. Then, **'move'** the microprocessor's oscillator setup (Q1, C9, C10) also as close to the microprocessor as possible.
- **'Move'** the voltage regulator's decoupling capacitors (C1, C2, C5, C6) as close to the voltage regulator as possble.
- **'Move'** the connectors (DCJ0202), headers (SV1, SV2), and switch (S1) last.
- Experiment for fifteen minutes with moving components around to minimize airwires.

When you understand how to place components, you can use your own arrangement of components, or use *part2.brd* from the downloaded lesson files to get an optimized arrangement.

6. Connections and Routing

Before we start routing airwires - or copper "traces", when they are physically laid out on the board - we will place polygons on both the top/red (layer 1) and bottom/blue (layer 16) of the board and connect them to the GND net. Polygons are large areas of copper (like large traces) useful for connecting a large amount of components together with a common node, such as the board's ground.

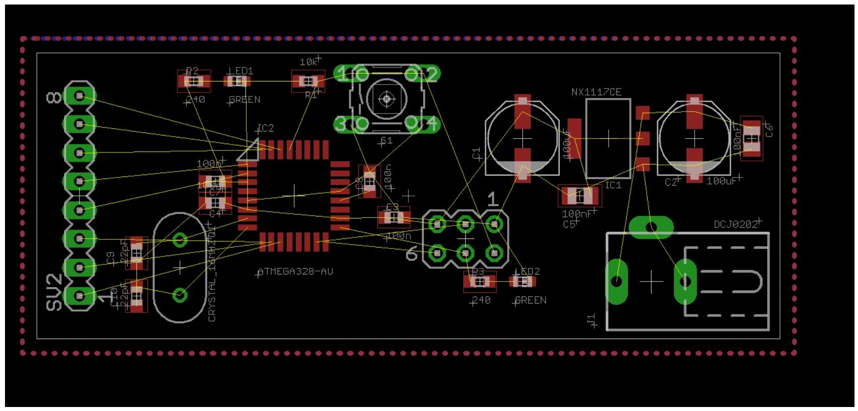


- Use the **'poly'** command or click the  icon to bring up the polygon tool. Then, click and create a polygon around the outline of the board.
- Click in the top-left corner and change the layer from "Red - 1 - Top" to "Blue - 16 - Bottom", like in the picture to the right.
- Then, use the **'poly'** command to create another polygon. This one will be on the bottom layer, or bottom, of the board. The board should now look like the picture on the next page (other than that your components may be in different places).

Now, we will tie these polygons to the GND net, using the **'name'** command (similar to Part 1, when changing the name of various wires).

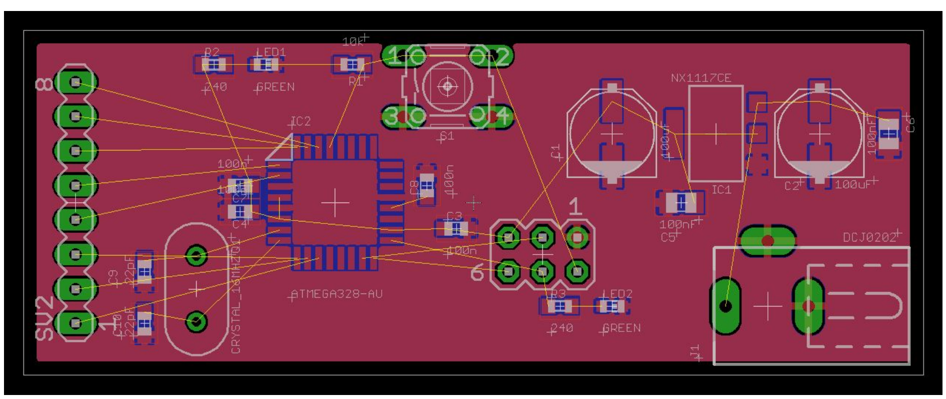
- Use the **'name'** command or click the 

icon, then left-click on the red/top polygon. (*You may need to RIGHT-click twice to switch between the red and blue polygons - a feature that EAGLE has to precisely select objects that are close together.)*



- Set the New Name field of the dialog box to be "GND", and ensure that the option "Change Name of this Polygon" is checked. Then click "OK".
- Use the **'name'** command to also set the name of the blue/bottom polygon in the same way.

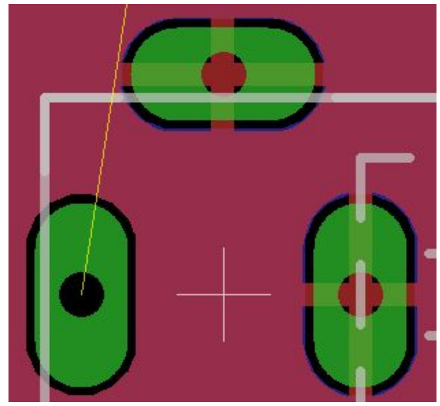
The **'rats'** command "fills" the polygon, so that way it fills in all available space and connects to GND airwires. The **'rats'** command also redraws airwires that are a part of the GND net, stitching them together with the polygon. Try it! It should look like the picture to the right.



Two other things are important to note from this picture. First, the polygon does not go to the edge of the board. This is because Eagle already has the minimum distance to the edge that the typical board house allows.

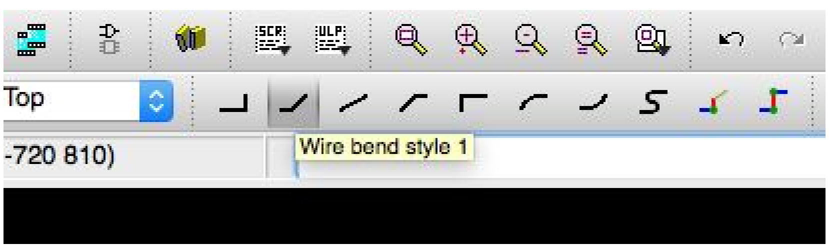
The last thing to note from this view are the "thermals". If you look closely at the aforementioned GND pins or the GND pin of the programming header (enlarged below) you will notice that the GND plane doesn't connect to the hole all the way around.

Instead, four small copper areas connect to the rest of the GND plane. This allows the pin to be soldered without the iron's heat being wicked away to other parts of the board through the polygon, potentially causing damage from the heat. The thermals also allow more of the heat to concentrate in the pin (also called a *via* when there is a pin that goes through a hole - marked in green).



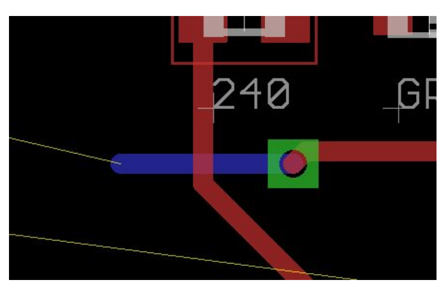
Now, you will practice routing the airwires on the board. Ask your MESH instructor if you need help.

- Use the **'rat; rip @;'** command to ripup the polygons and put them to their former border-only selves, so the board is visible for routing.
- Use the **'route'** command or click the  icon. Then, select the first "Wire bend style 1" from the menu above the command line, as to the left picture (feel free to experiment with the other ones, but 45-degree angles are typical for electrical noise and historically, ease of manufacturing). Then click on one of the airwires to begin routing it. Click down on the board to partially route a trace, then continue routing. Click in the center of a terminating pin or pad to finish the trace (the airwire will be gone to let you know this occurred).

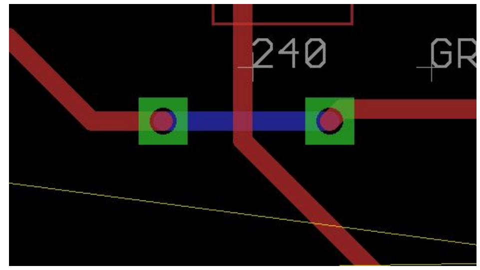


If there are two airwires that are overlapping and you cannot be creative with routing the traces around one another so they do not intersect, you can use a via to dip underneath the first top layer and onto the bottom layer temporarily to avoid an intersection.

- While using the **'route'** command, click down before the intersection to anchor the wire. Then, switch to the bottom layer using the



layer picker in the upper left corner. Then, route as little as necessary on the bottom layer to avoid the intersection, click to anchor the bottom layer trace, and then switch to the top layer to finish the trace (as to the picture on the left). Look at the picture of the right to see a finished trace.



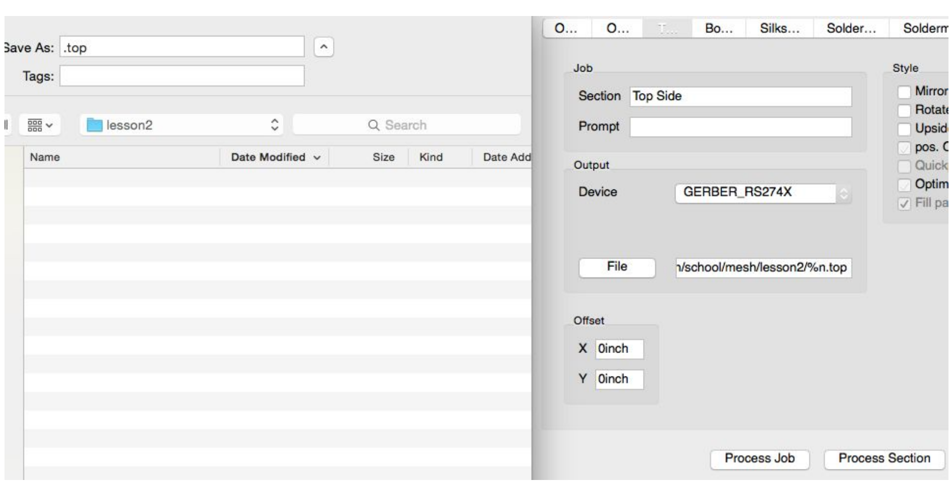
7. Gerber Files

Once you have made the board in Eagle, you need to specify to the board house what the PCB looks like, using Gerber files for specifying 2-dimensional PCB layouts- an open source file format standard. To do this, we use Eagle's CAM processor.

We will also go through the process of downloading a PCB fabrication house's CAM job, and processing our board design with that. We will use Sunstone Circuits' CAM jobs.

- Please download the located “Sunstone-EAGLE_5.0_cam_NEW” file at (http://www.sunstone.com/pcb-resources/pcb-downloads), and unzip the file after it has been downloaded.
- Copy this folder to EAGLE's CAM folder, located wherever the EAGLE application is installed (for Windows, in C:/Program Files (x86)/EAGLE...)
- To open the CAM processor, go to File → CAM Processor or press the  button.
- With the CAM window open and brought to the forefront, go to File → Open → CAM Job, navigate to your Downloads folder, to the Sunstone-EAGLE_5.0_cam_NEW folder, and open “2L-Plus-Sunstone.cam”

Now that the CAM job is loaded, each of the new tabs at the top represents a layer or collection of layers that are useful to the fabrication house, from the outline, to the top and bottom sides, to the “silkscreen” on the board.



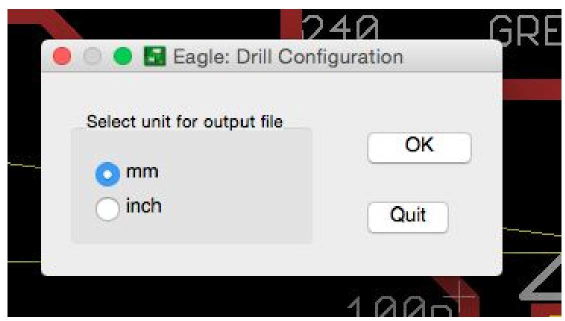
Create a folder on your Desktop named “output”. For each tab at the top change the “File” parameter to include the path to the new folder 'output', by clicking the “File” button and finding the correct folder. Don’t forget to remember the extension, and manually enter it in the “File” window (displayed on the left, in the picture on the right)! Once done, press the 

button to create the CAM files in the ‘output’ folder.

Next, we’ll create placement information for the holes on the board, with another CAM job.

- With the CAM window open and brought to the forefront, we’ll open another CAM job. Go to File → Open → CAM Job, and open “excellon.cam”.
- Save the file to the same place as the other Gerber files, and process the job.

The final step in the CAM process is to generate the tool list file. This file provides information to the board house about what size drills your job will need. For this, we will use a ULP, or a User Language Program - a script written with C-like syntax.



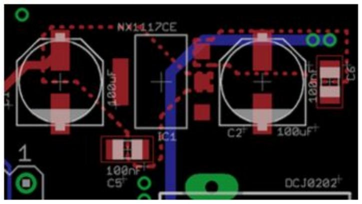
- Type the **'run'** command, and in the window that comes up, navigate to ulp → drillcfg.ulp.
- In the window that opens after opening “drillcfg.ulp”, specify the units as mm. Press OK.
- Press OK again to accept the sizes listed.
- Save the .drd file with the rest of your Gerbers in 'output'.

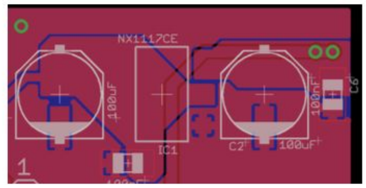
If you were to send your board to a fabrication house, you would compress the ‘output’ folder that contains your gerbers, and send them that file.

8. Self Check

For this self check assignment, finish routing the board started during the lesson if you have not already. The following images with their captions detail a few areas of the board and the key items to consider when working in these areas that were not yet covered in the lesson.

8.1 Power Supply

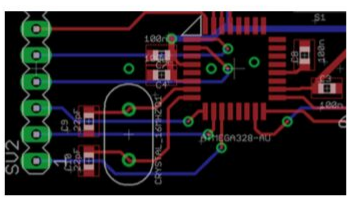


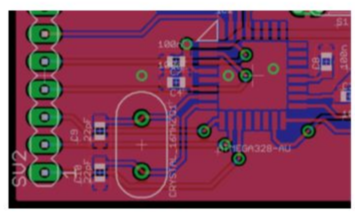


When working on the power supply, it’s important to keep traces wide. All the system current will flow through these traces so it’s best to make them polygons. As seen in the images above, there is a polygon for each of VIN and VOUT in the power supply. GND, as mentioned above, exists in a polygon that covers all uncovered areas of the board.

A special thing to note when working with overlapping polygons is the **rank**. To view or modify a polygons rank, use the **'info'** command and click on the polygon's outer edge. Polygons with higher rank (lower number) will be placed over lower ranked polygons. In this case, make sure the power supply polygons are filled instead of the GND plane polygon (have a higher rank, or lower number).

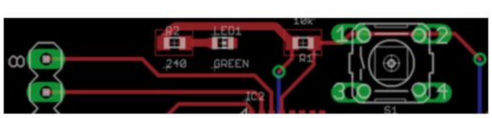
8.2 ATmega





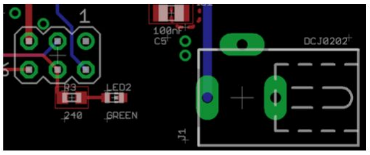
As seen in these figures, the organization of components is critical. Place decoupling capacitors as close to their IC as possible. The crystal is of secondary importance to those capacitors and should be placed very near the microcontroller as well.

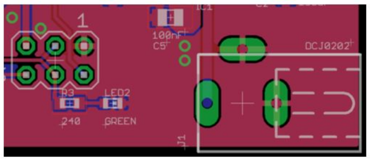
8.3 Button



The reset button can be placed anywhere convenient. Remember that in your design you will need to access to the reset button. In your design, double check that any buttons are placed so that you can actuate them. Don't forget that some components like electrolytic capacitors and connectors can be fairly large.

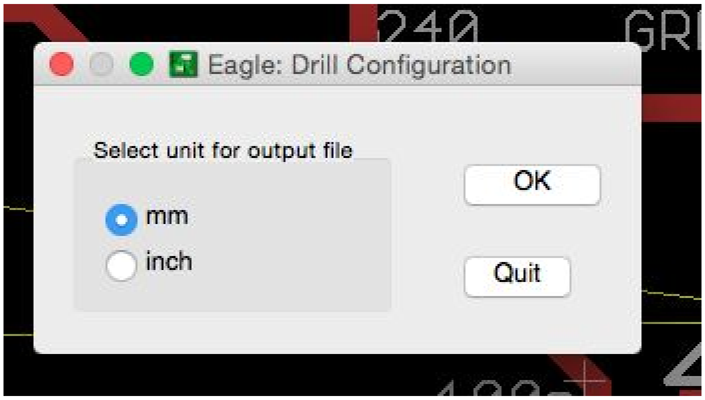
8.4 Connectors





Like the button, external connectors are positioned where the can be accessed for regular use. Here the bullet power connector is at the board edge with the plug pointing out to the edge of the board. Double check that all your physical connectors do not conflict with other mechanical parts of your design.

9. OPTIONAL PART: Producing stencils



There are times when the complexity of surface mount items on your board becomes too high to solder by hand without assistance. For these situations, a solder stencil helps simplify the process of accurately placing solder paste on each component's pad.

Open the stencil tutorial by clicking the link below:

http://www-personal.umich.edu/~sdebruin/Producing_Custom_Paste_Stencils_1v1.pdf

Follow the guide to produce a .dxf file from your board. A .dxf file can be used on a laser cutter to cut stencils in plastic sheets. If you would like to get physical stencils made for a project on a laser cutter, please talk to your lab instructor.