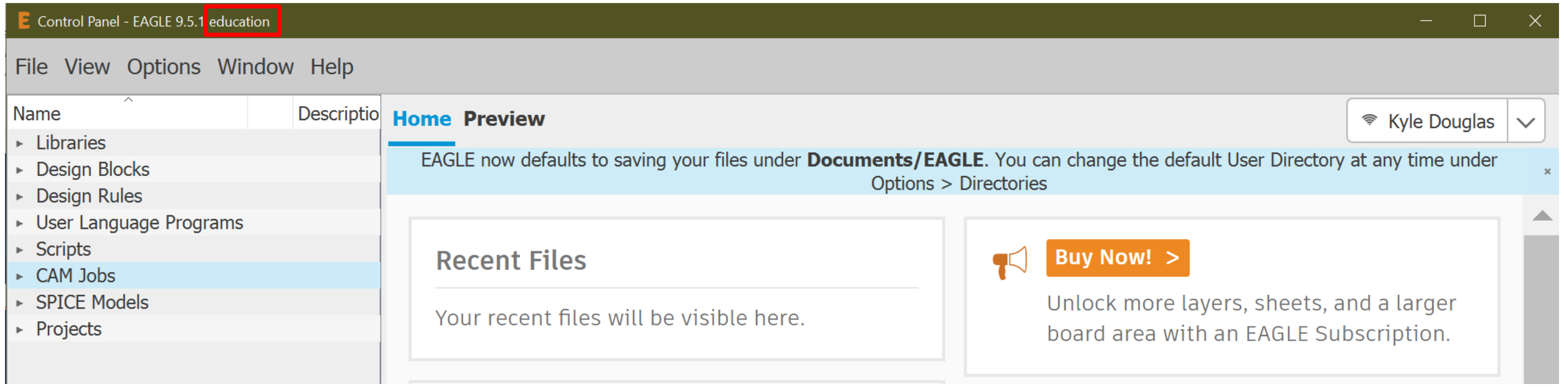
Creating a PCB

ECE 189A

OCTOBER 4, 2019

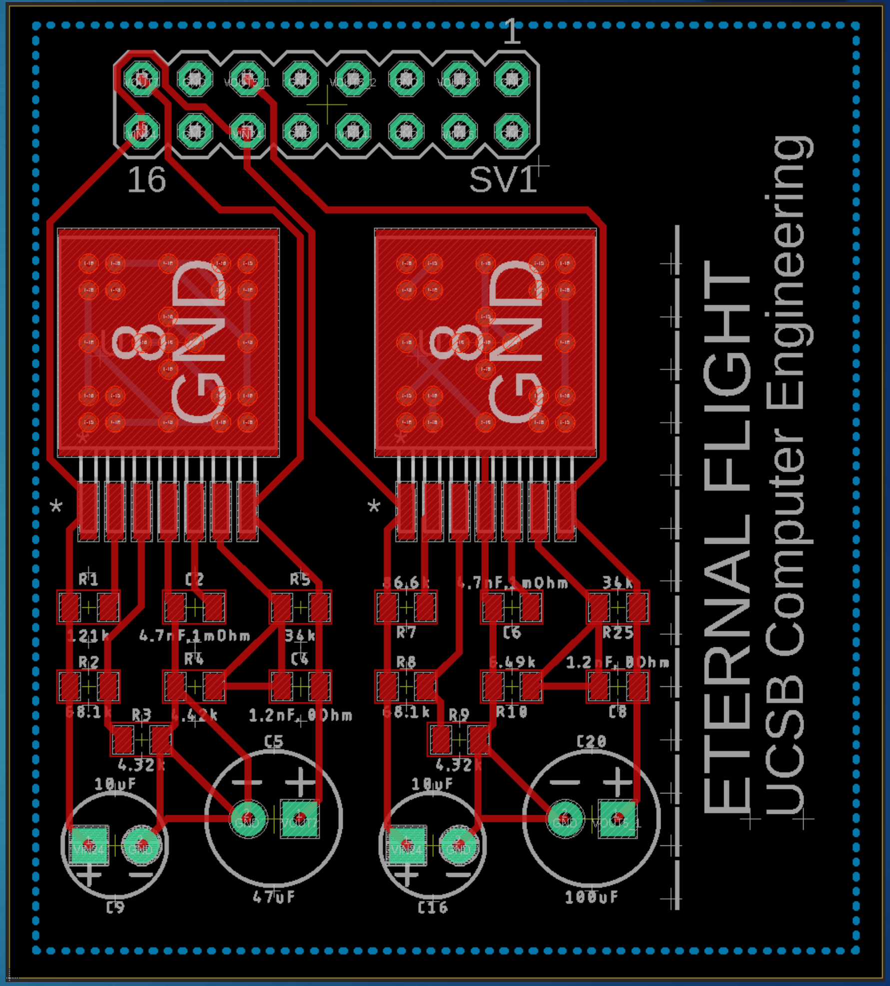
Create an Autodesk Account

- Follow the link to create an account
- Enter relevant info
- Update your account for education: link (Note: This may take some time, so do this at least a day before you start the homework)
- Download Eagle for your OS: link (Not needed if you are using the lab PCs)
- Sign into your account on Eagle, make sure the license in the top left corner says ‘education’



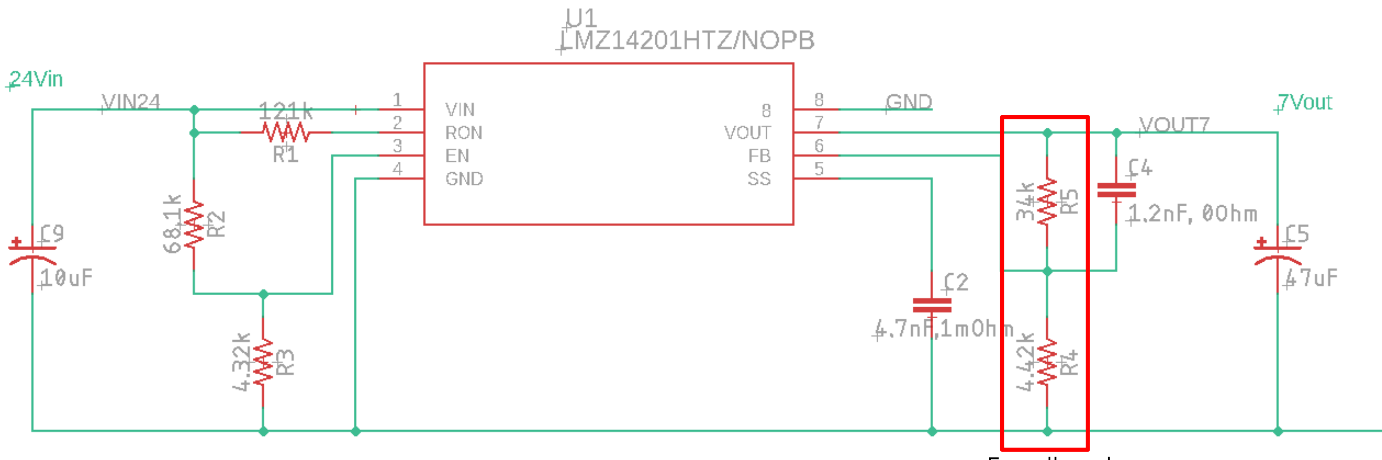
PCB Creation Example

- The following PowerPoint details the creation of the power distribution board for one of the drones used in Eternal Flight
- The goal of the circuit was to accept 18-28V input and output steady 5V and 7V sources
- This was accomplished using 2x TI Linear Switching Module LMZ14201, one for 7V, one for 5V



PCB Creation Example

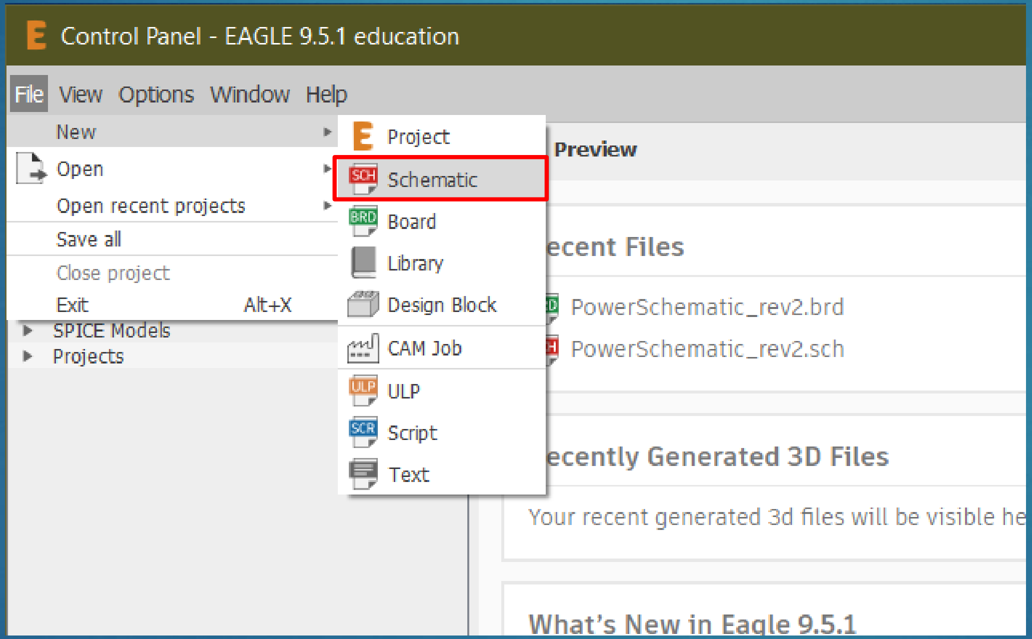
- One voltage regulation circuit was repeated twice to supply the two different required voltages
- The main difference between the two circuits is different feedback resistor networks
- These values were selected carefully based on the datasheet of the switching regulators



Feedback Network

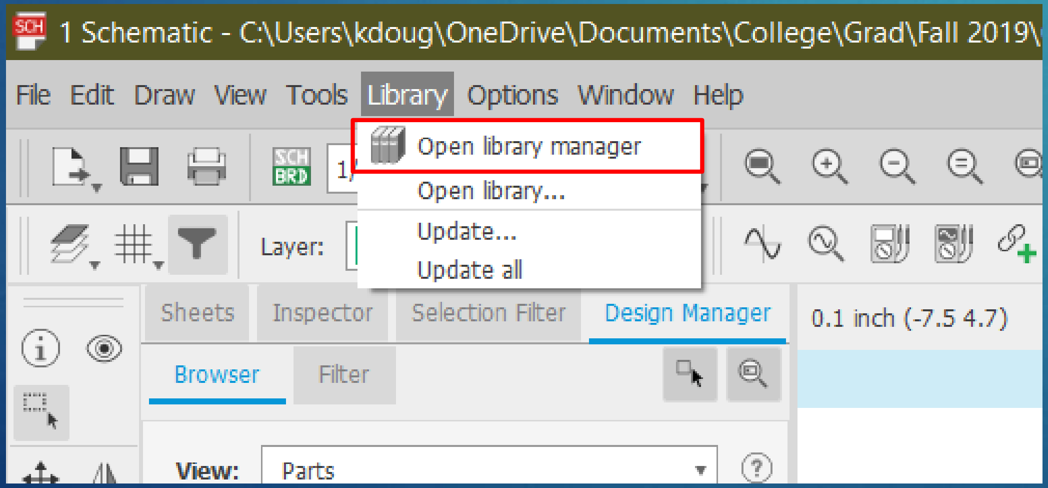
Create New Schematic

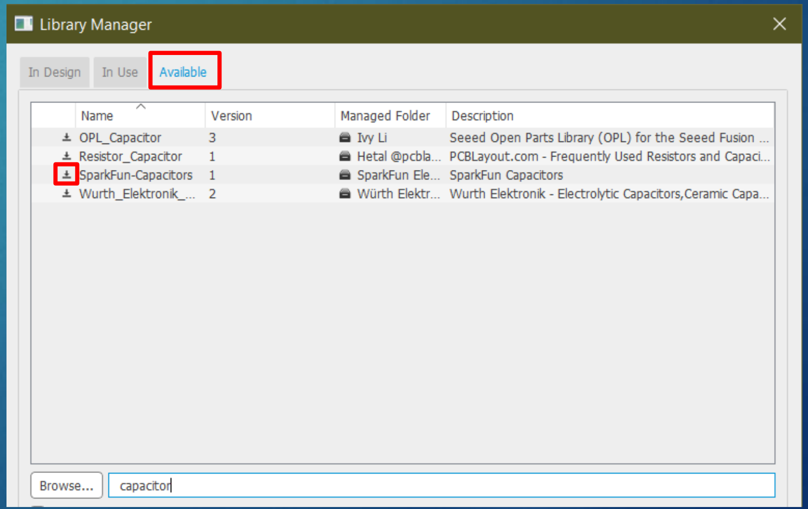
- On the main control panel, select File->New->Schematic



Add Libraries

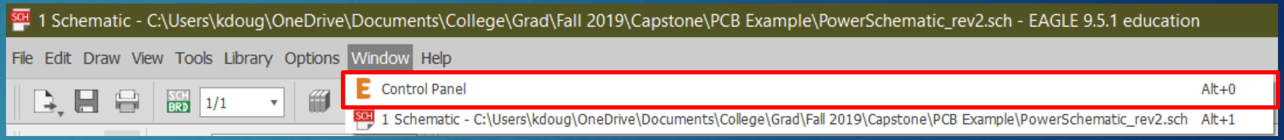
- For Custom Libraries:
  - Copy files with '.lbr' extension to to C:\Users\USERNAME\Documents\Eagle\libraries
  - Restart Eagle
- Many Libraries are available and can be easily imported through Eagle's tool:

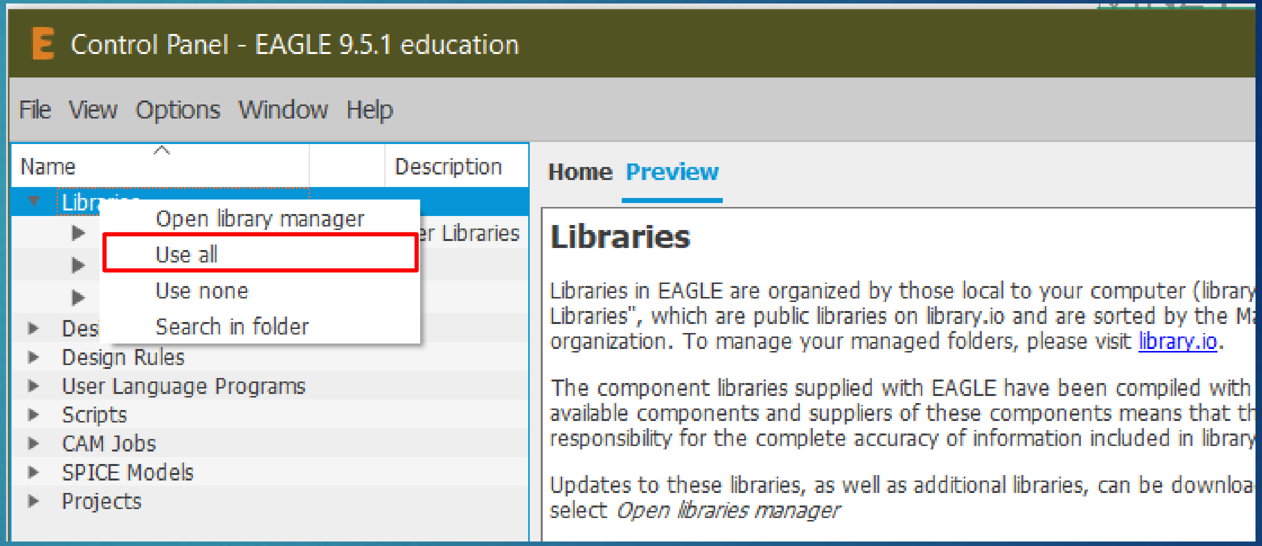




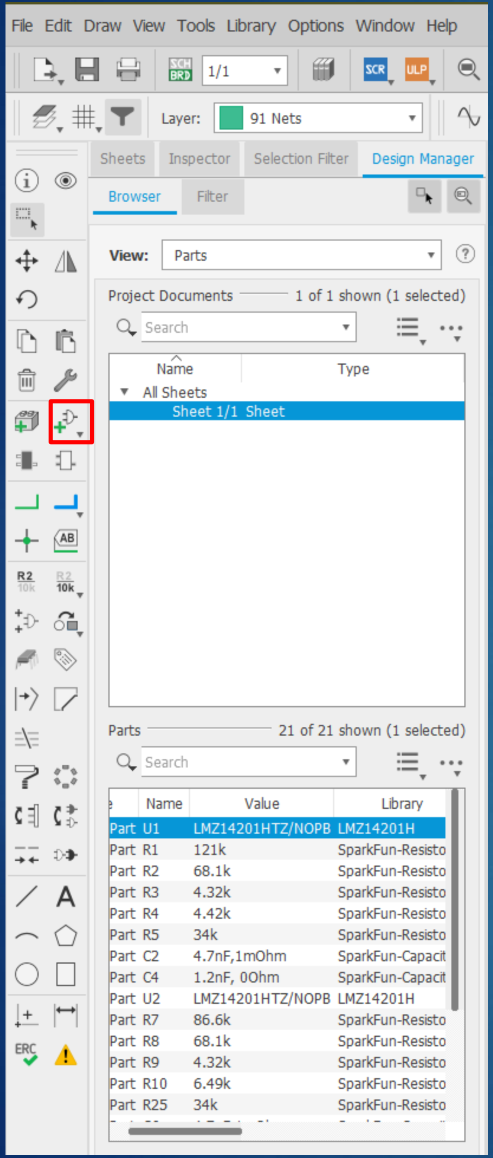
Enable Libraries For Project

- Open Control Panel
- Right Click Libraries
- Select Use All
- Note: Alternatively, only select the libraries you intend to use on this project

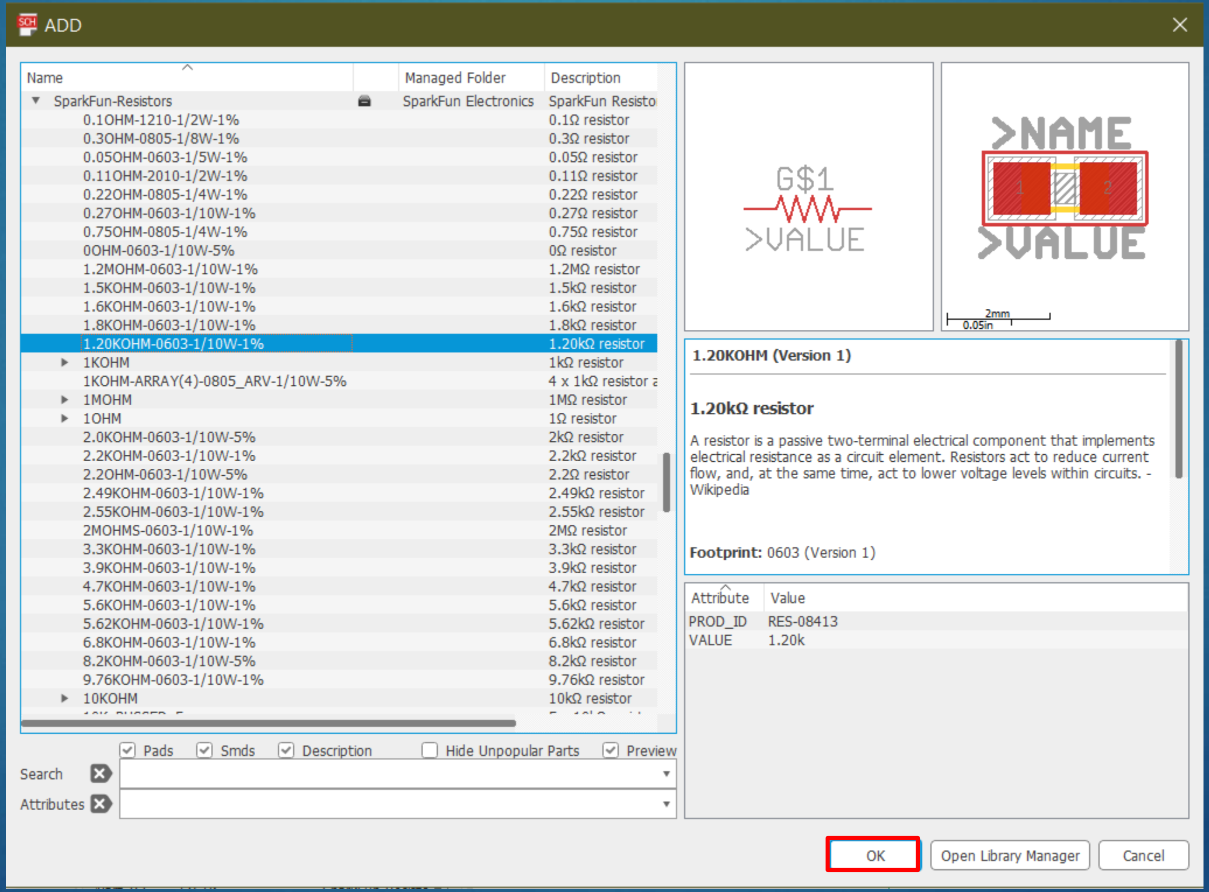




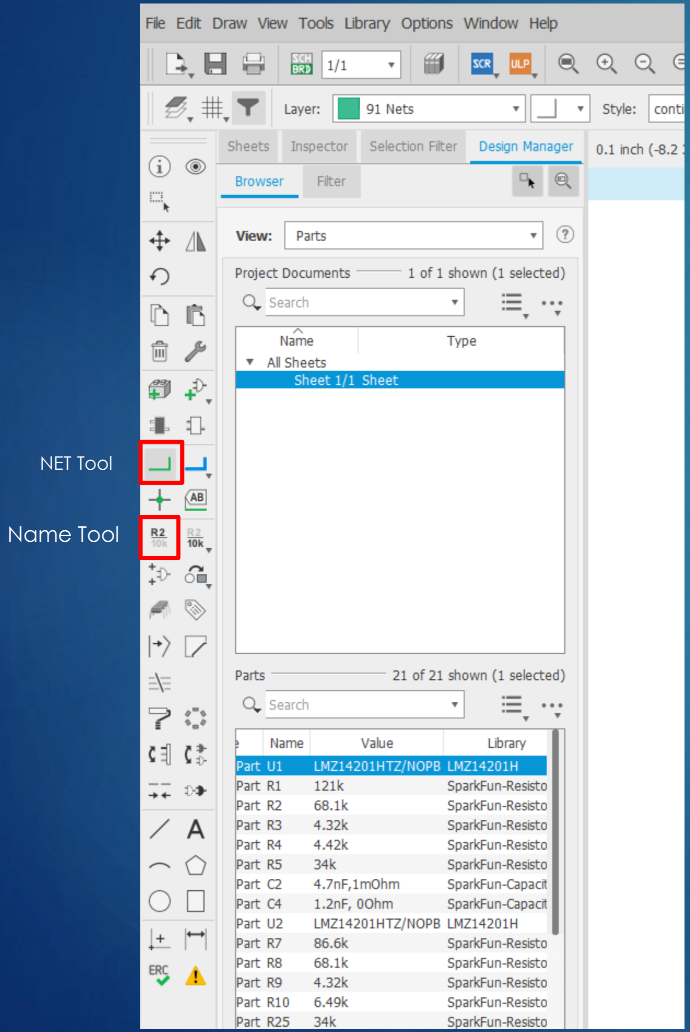
Insert Parts



- Find your desired part in the ADD menu
- Press OK to place into your schematic



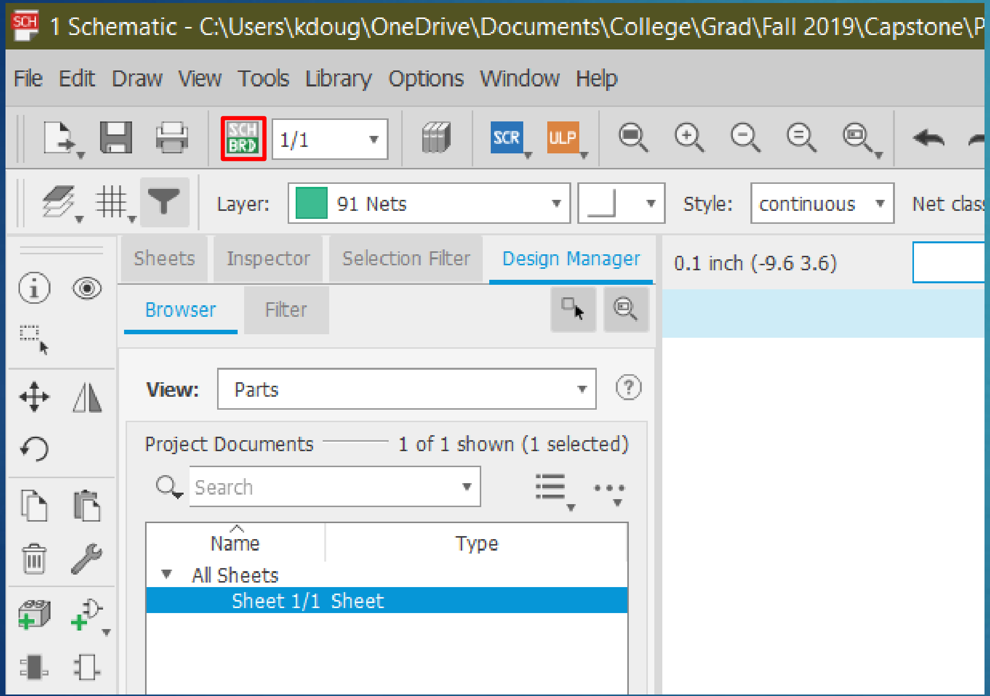
Make Connections

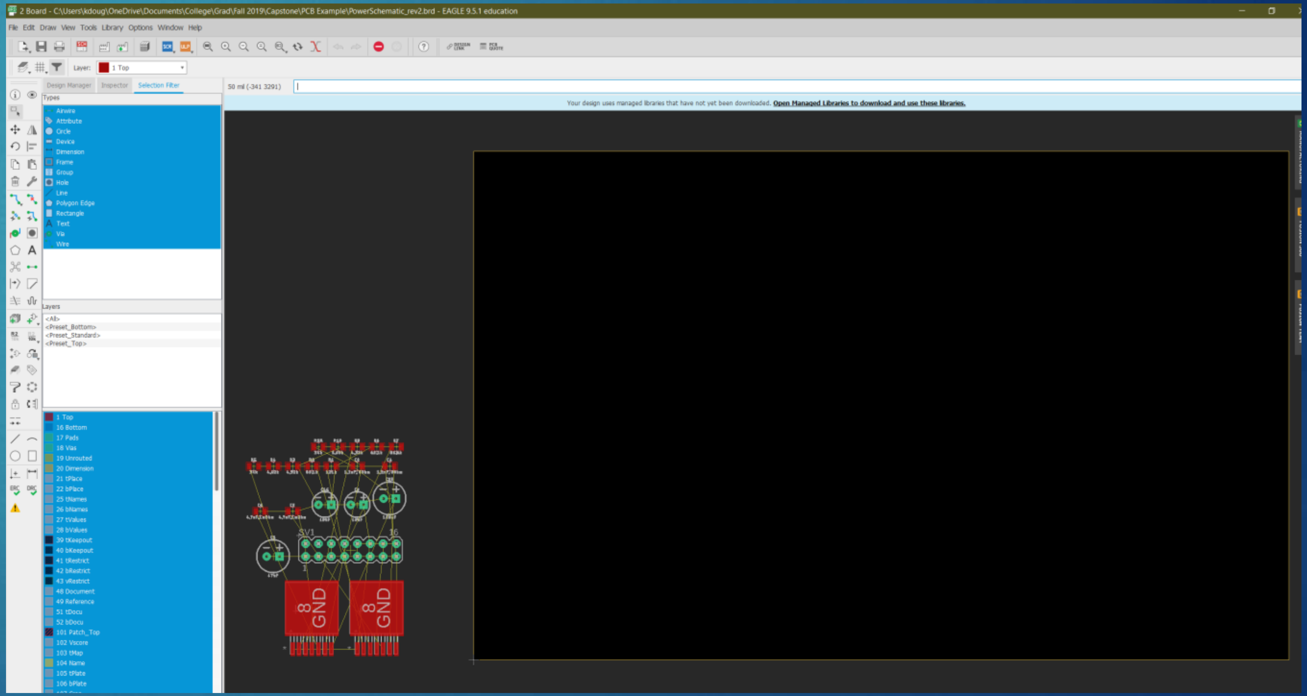


- Select the NET tool, and click to place wires
- Note: Be careful to make sure that the net locks onto each part you connect it to. A missed connection is typically one of the biggest causes of problems in PCB design
- Useful Tip For “Wireless” Connections: if you use the NAME tool, and name two nets the same, Eagle will assume they are connected, even without an explicit wire.

Begin Routing PCB

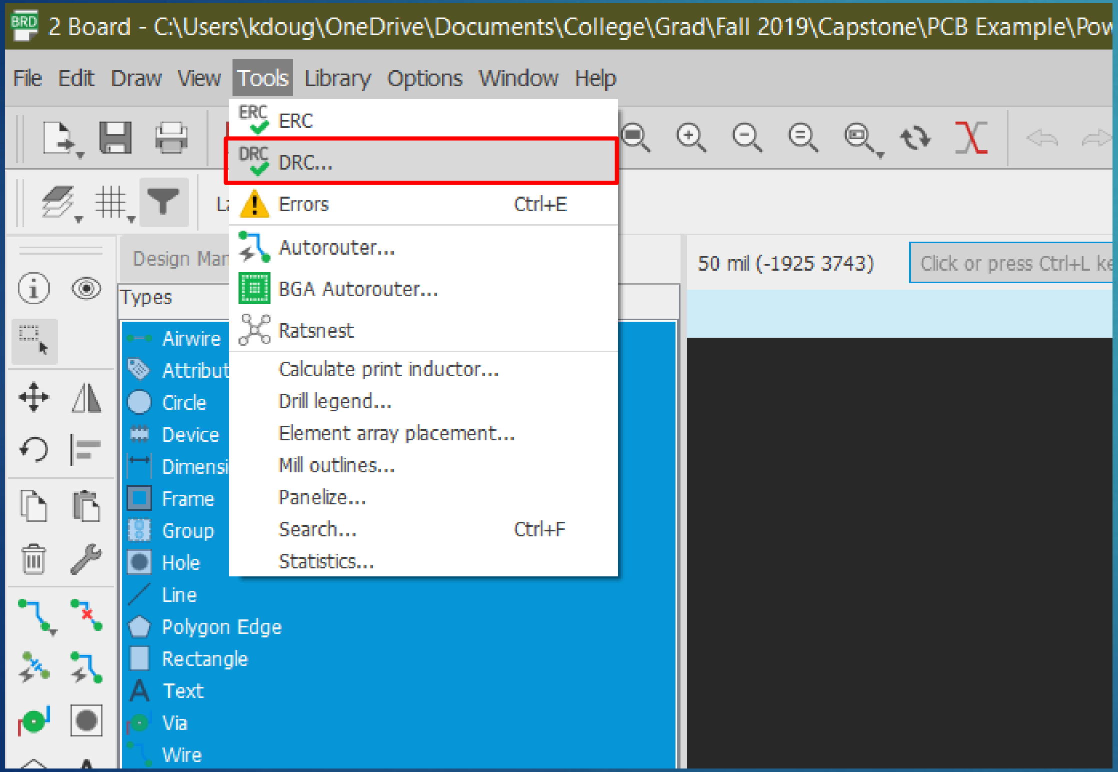
- Once your schematic is complete, press the BRD button on the Eagle Toolbar
- A new Routing Window Will Appear

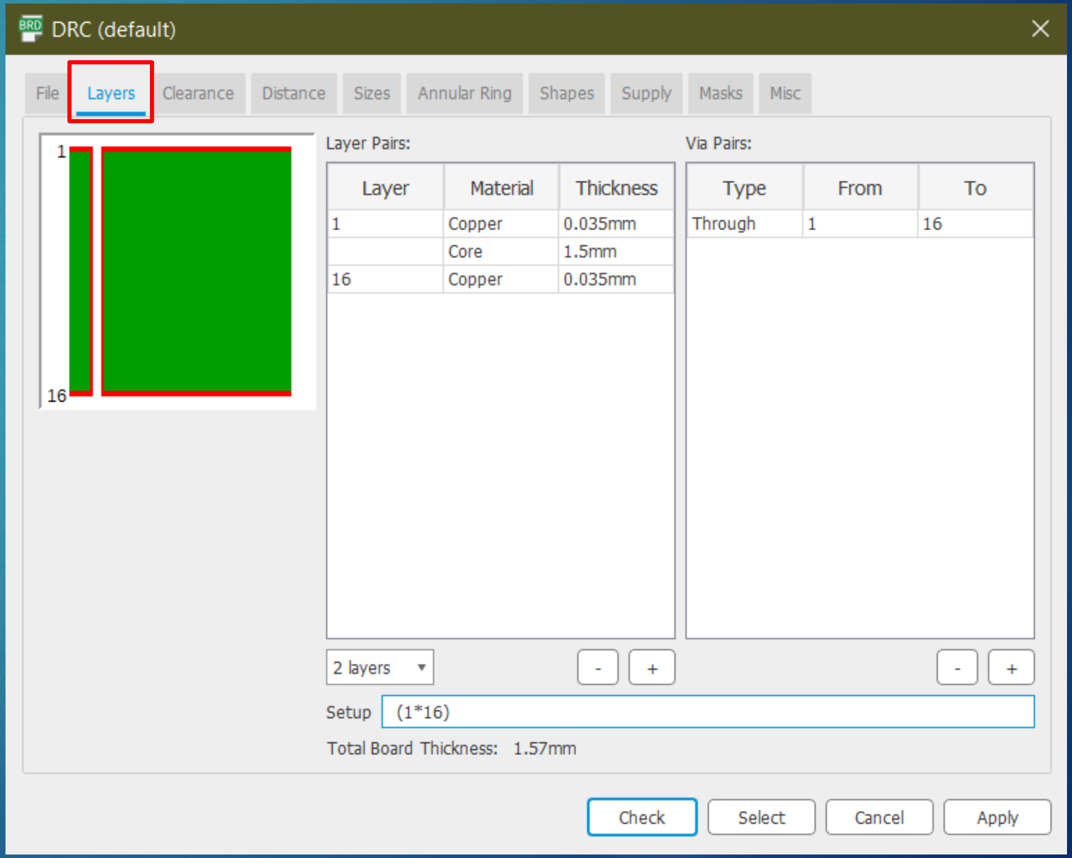




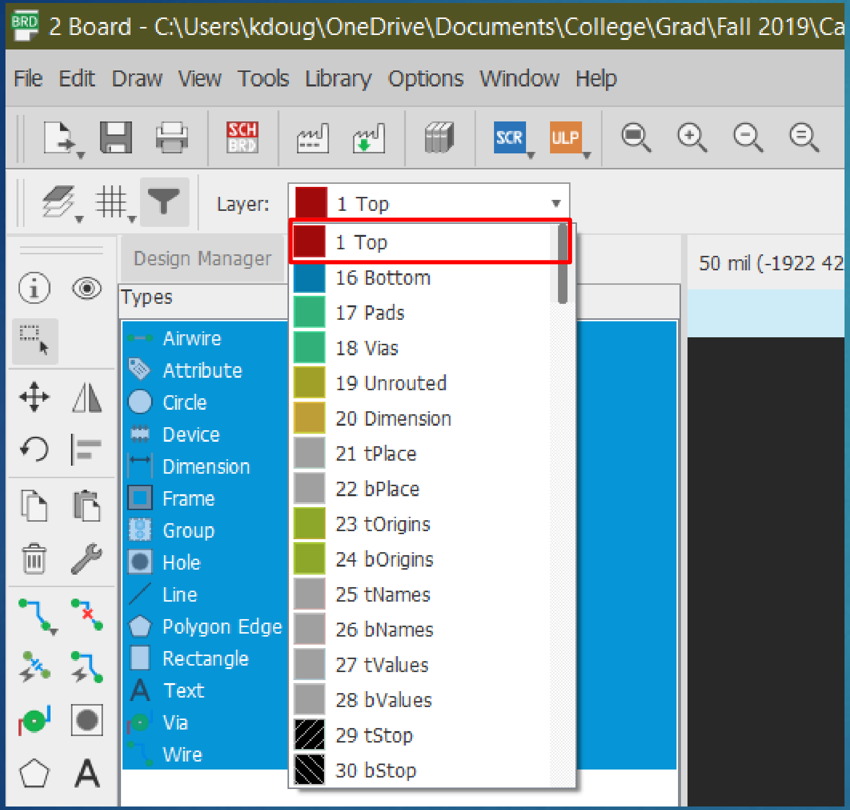
Define Layers

- Layers are added in increments of 2, where the first half of the layers start from the top and the second from the bottom. For example, for a 4 layer board, enter (1*2*15*16) in setup
- Each layer is separated by a ‘core’ layer to prevent shorts

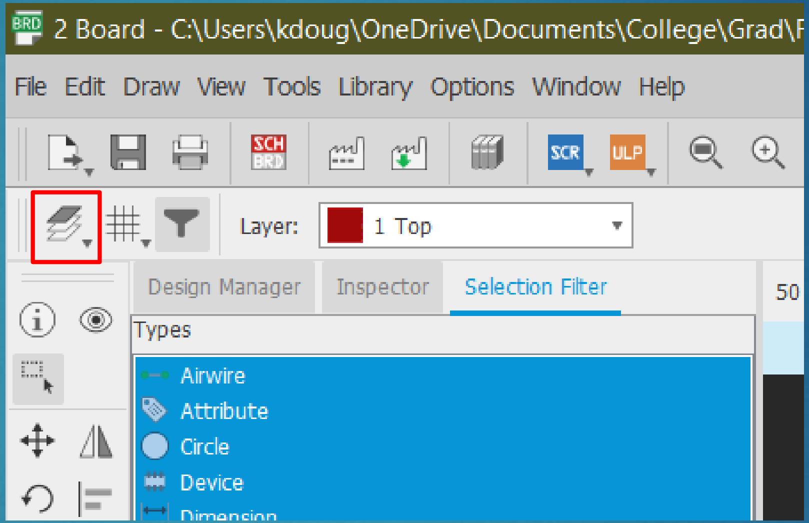




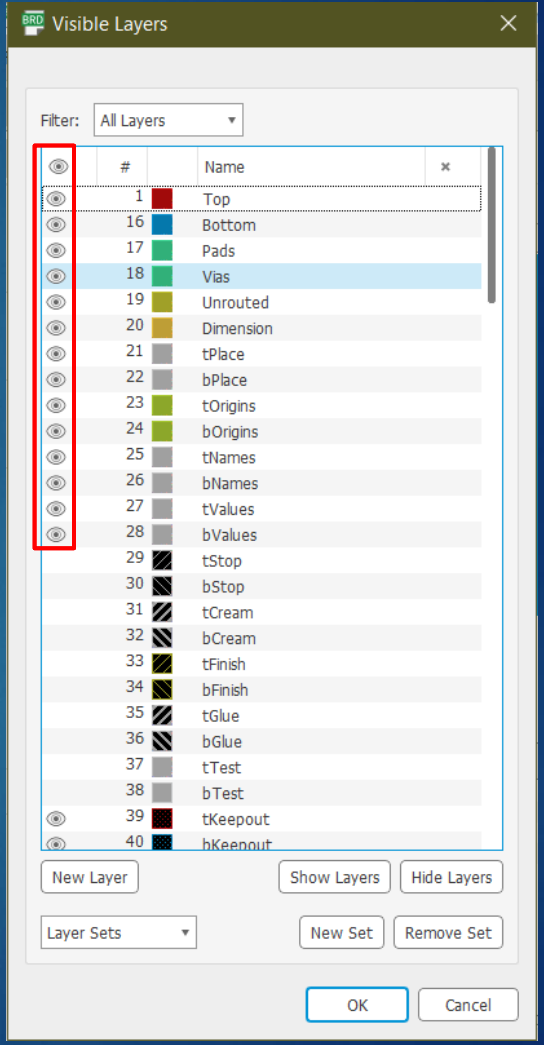
Move Between Layers and View Layers



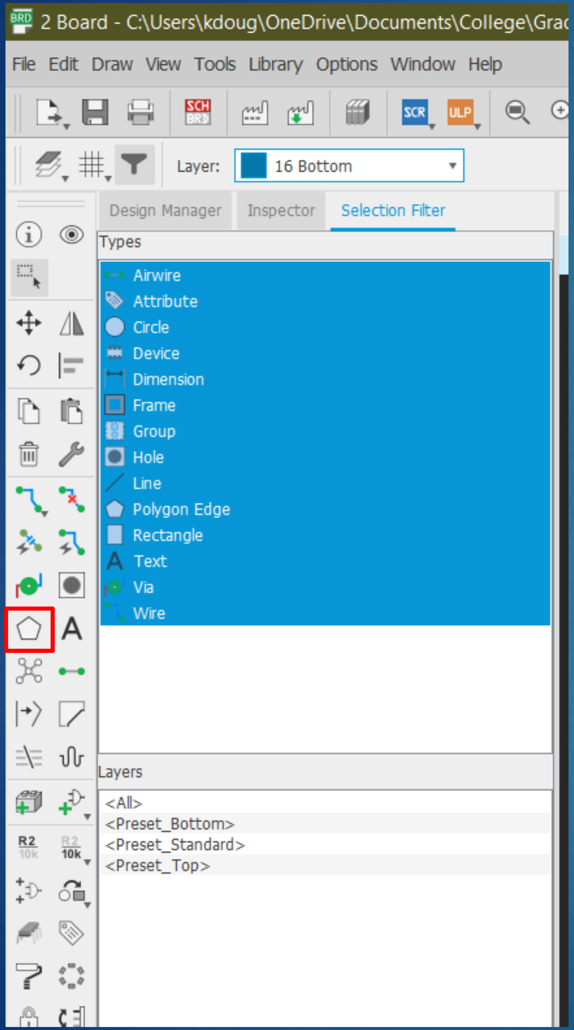
- Select Active Layer

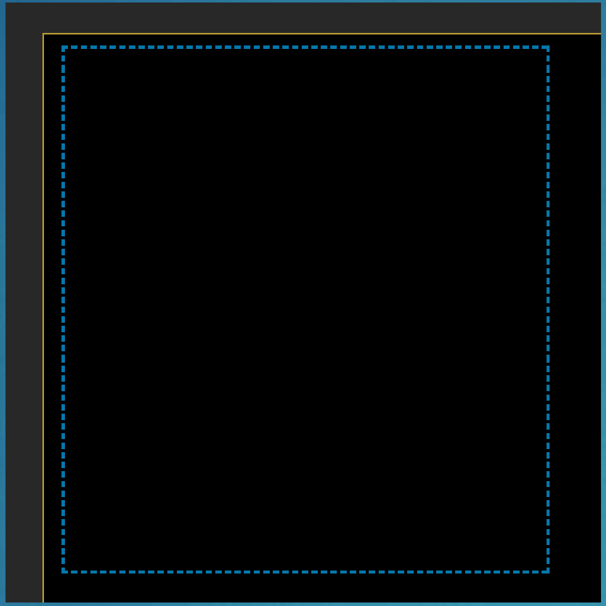


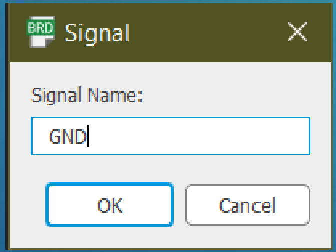
- Toggle Layer Visibility



Add Ground Plane

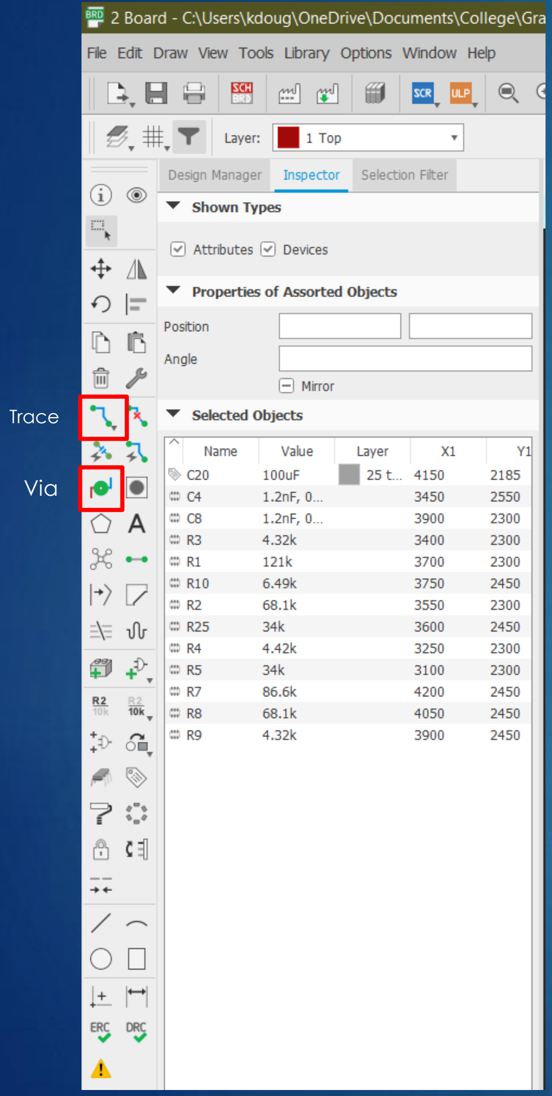




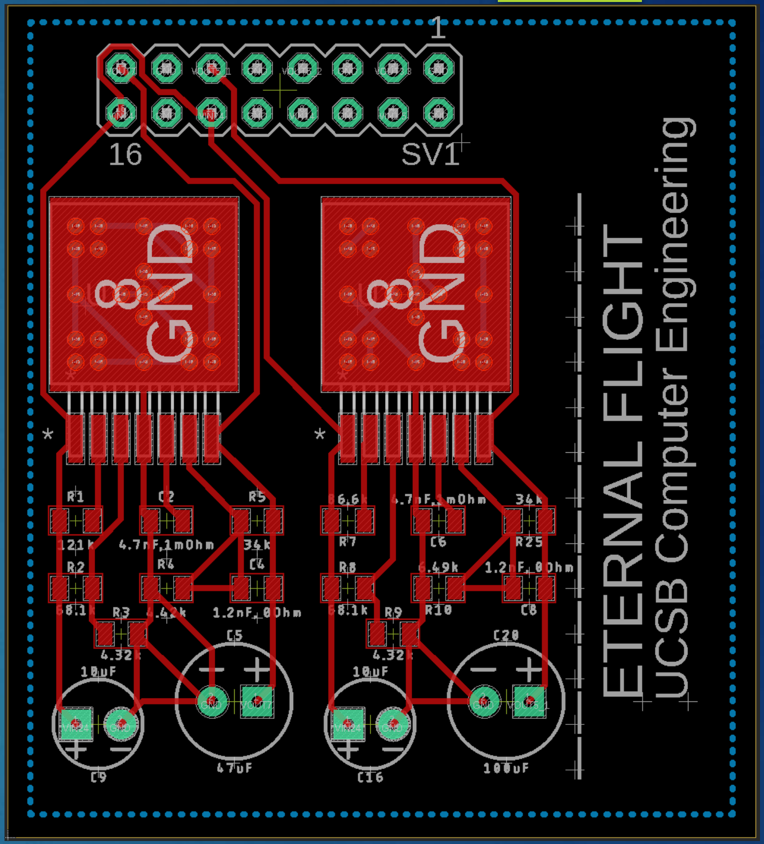


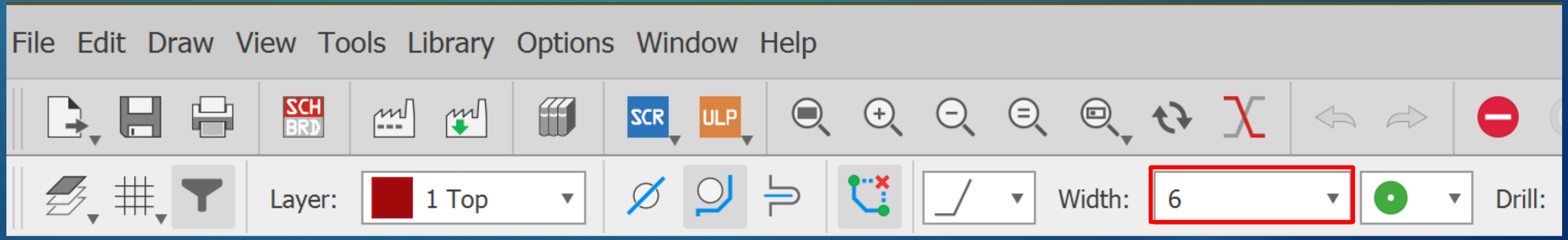
- A ground plane is a section of solid metal on your PCB that is shorted to ground. This idea can be used for any constant voltage, but is extremely useful for ground since most circuits have numerous connections to ground
- More info here
- Select the Polygon tool from the menu
- Draw the Shape of your ground plane
- Name the signal with the same name as the ground net in your schematic
- Use Tools->Ratsnest to fill in the ground plane

Place Components



- Arrange your components in your desired layout
- Add Traces to make connections
- Note: Higher width traces reduce resistance and are better for high current applications. In most cases the default trace is sufficient
- Add vias to move between layers
- Note: Never use greater than a 45 degree angle for traces. This adds resistance and interferes with signals.

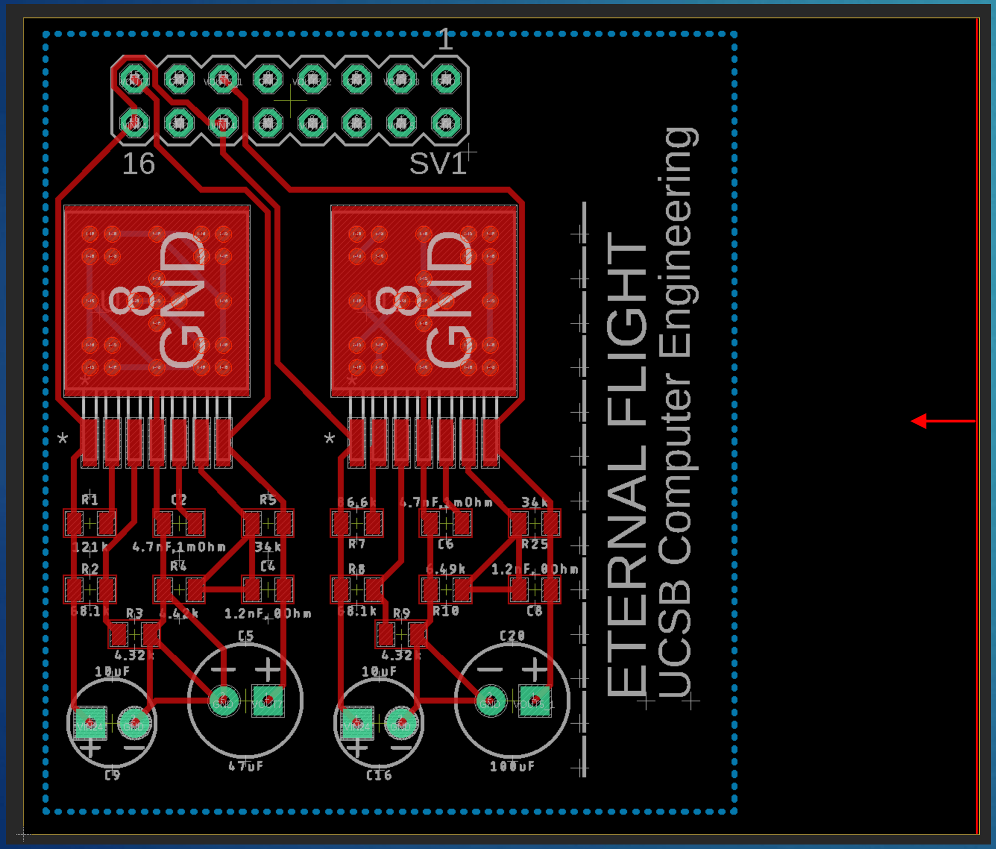


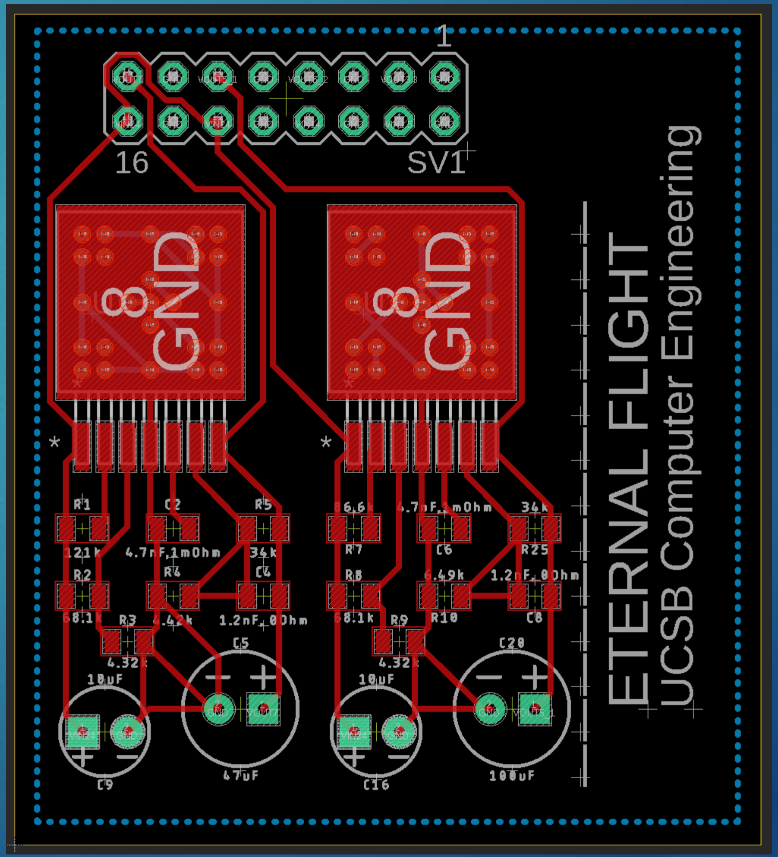


Trace width

Define Edges of PCB

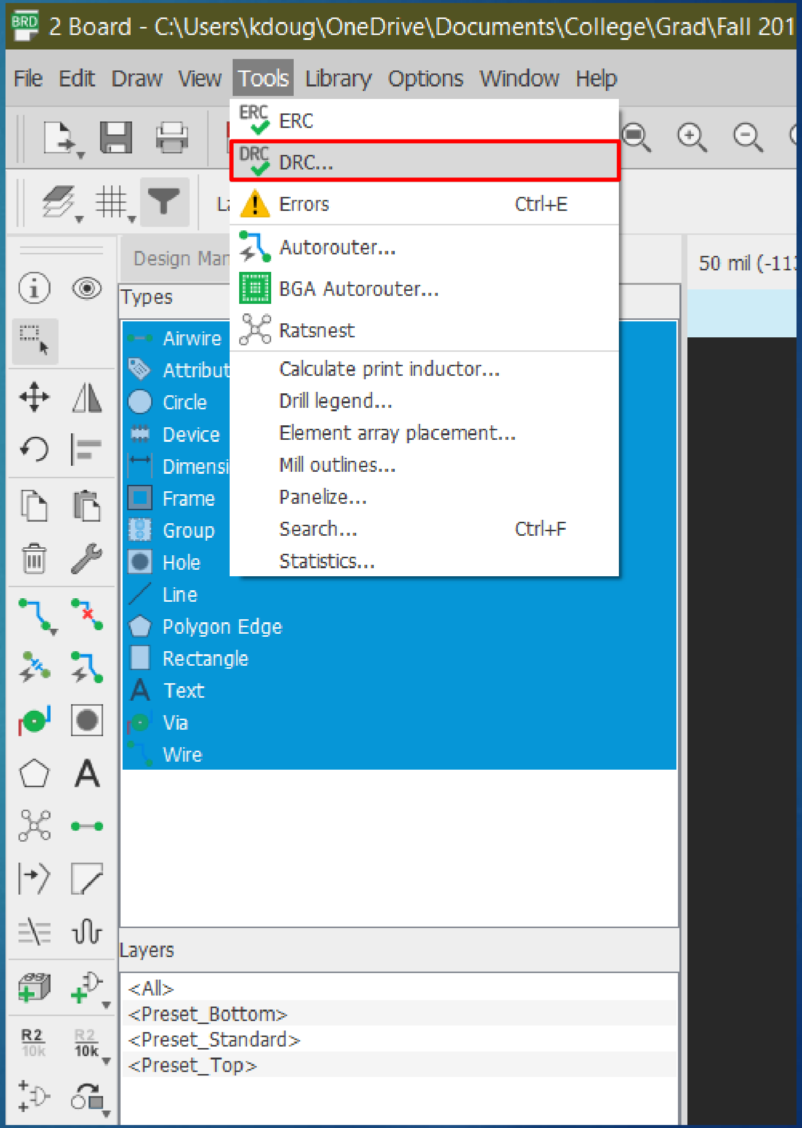
- Click and drag yellow edges to create desired shape around inner components





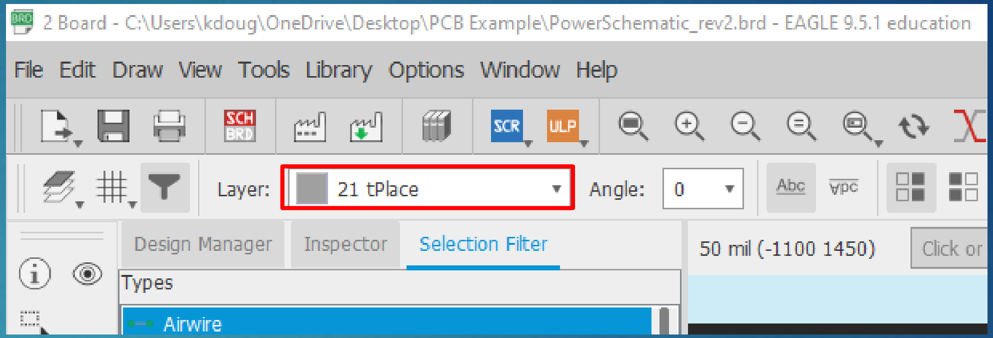
Design Rule Check

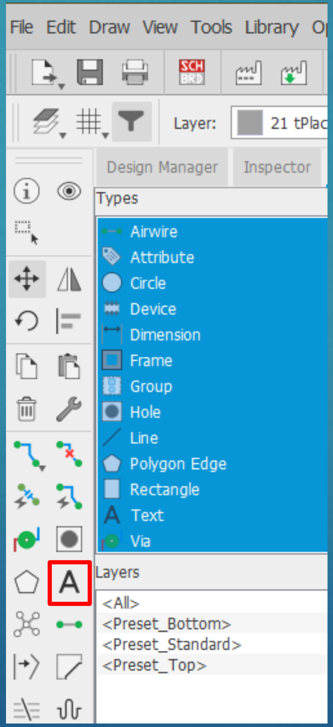
- Select Tools->Design Rule Check
- Select Check
- Correct any errors that appear

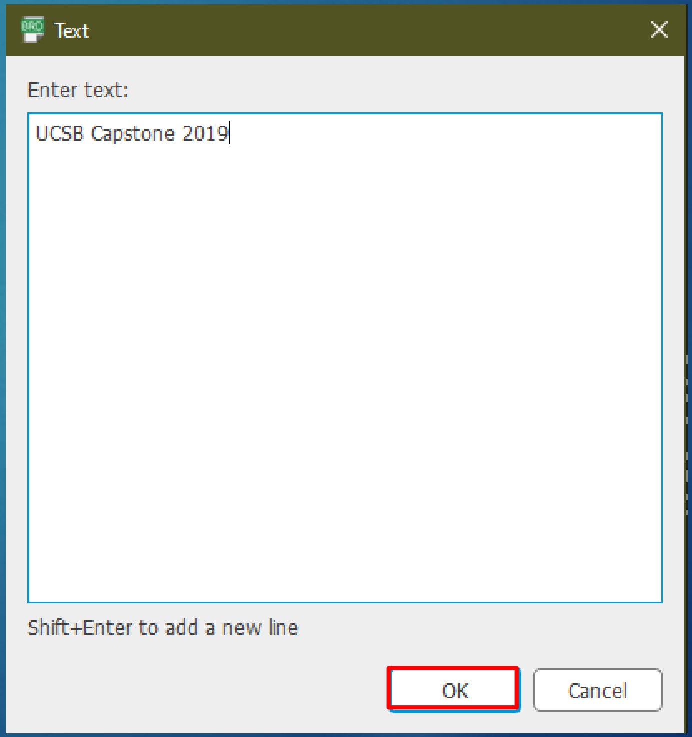


Add Silkscreen

- Select the layer labeled tPlace
- Select the Text tool
- Enter Desired Text
- Place on PCB

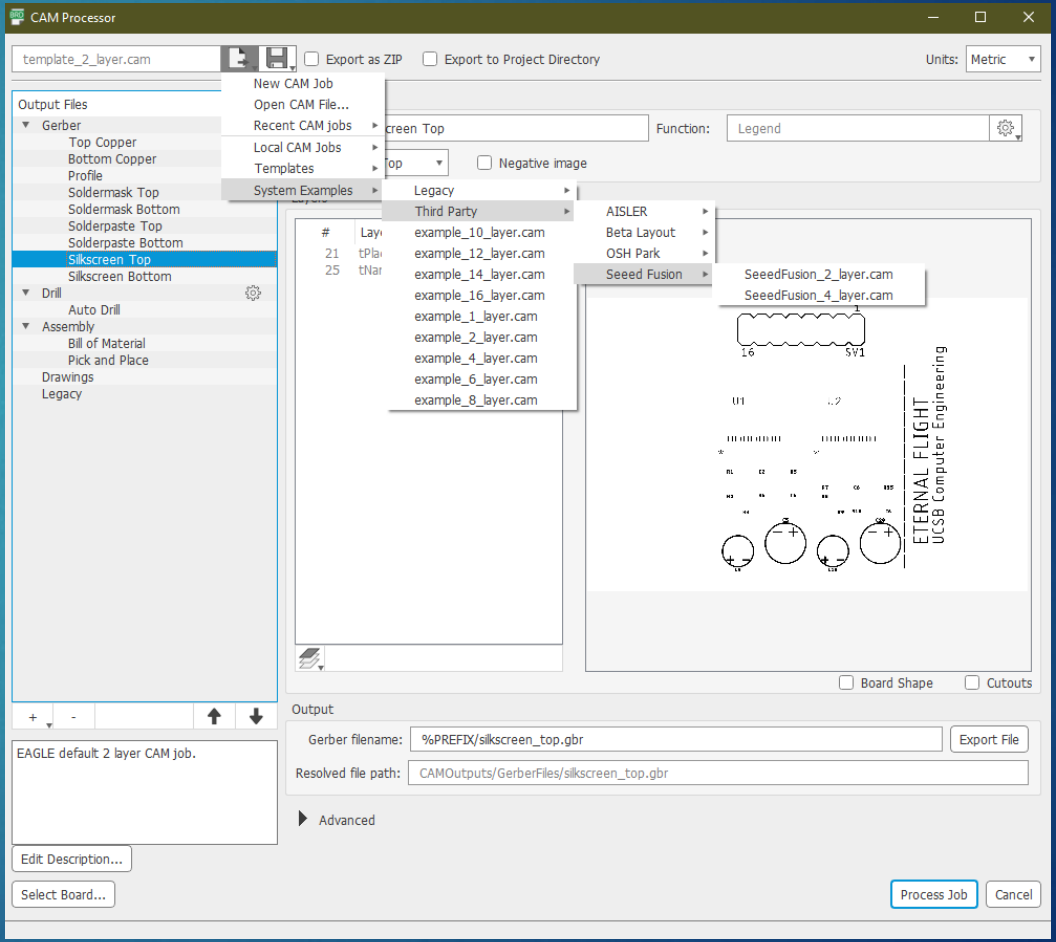




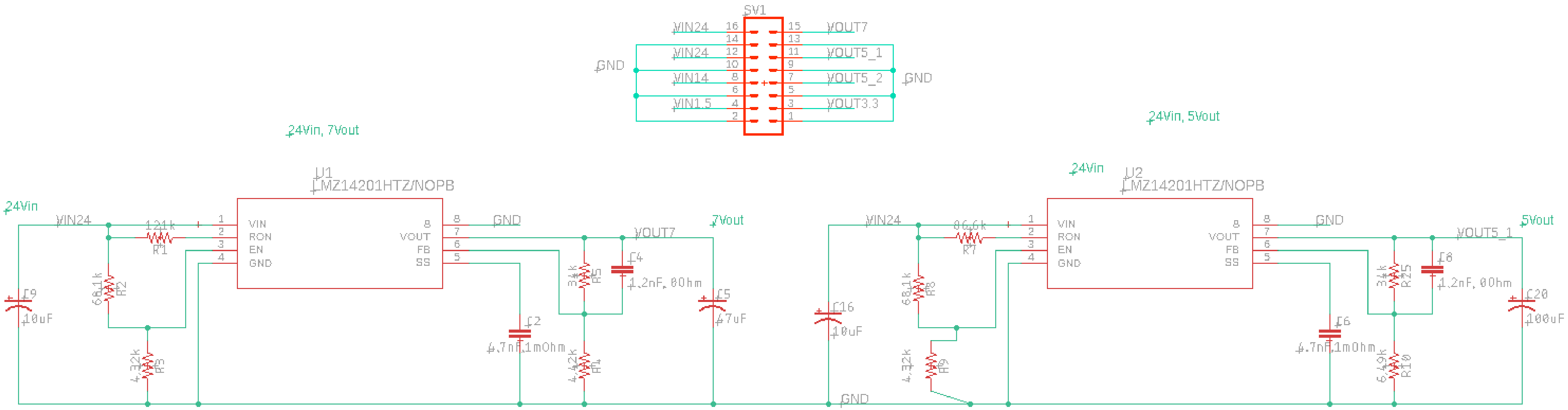


Create Gerber Files to Send to Manufacturer

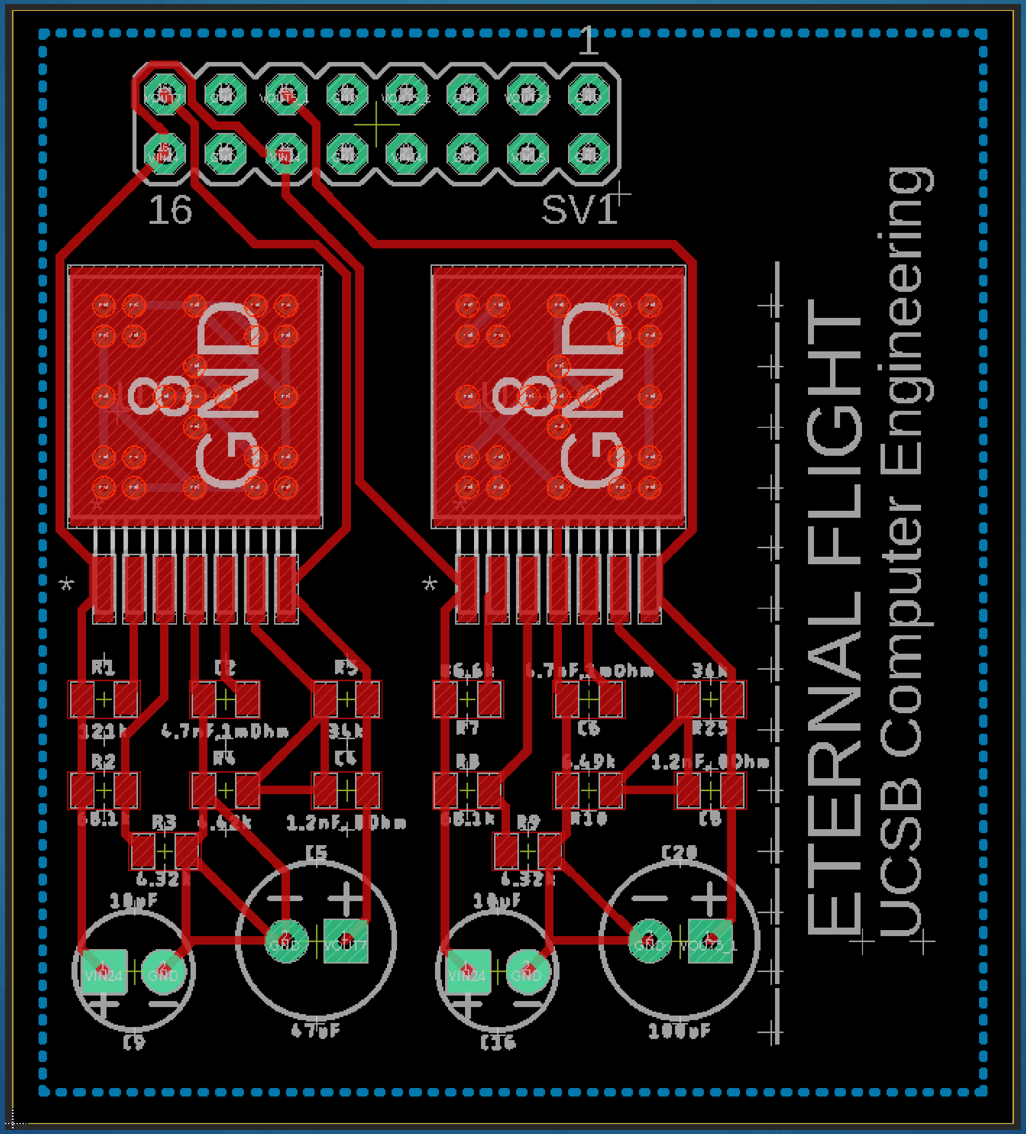
- Open the Cam Processer with File->Cam Processer
- Select the correct config file for your board
- System Examples -> Third Party -> SeeedFusion -> SeeedFusion_X_layer.cam
- Select Process Job at the bottom right of the window
- Check the output here, simply upload the zip file generated by Eagle



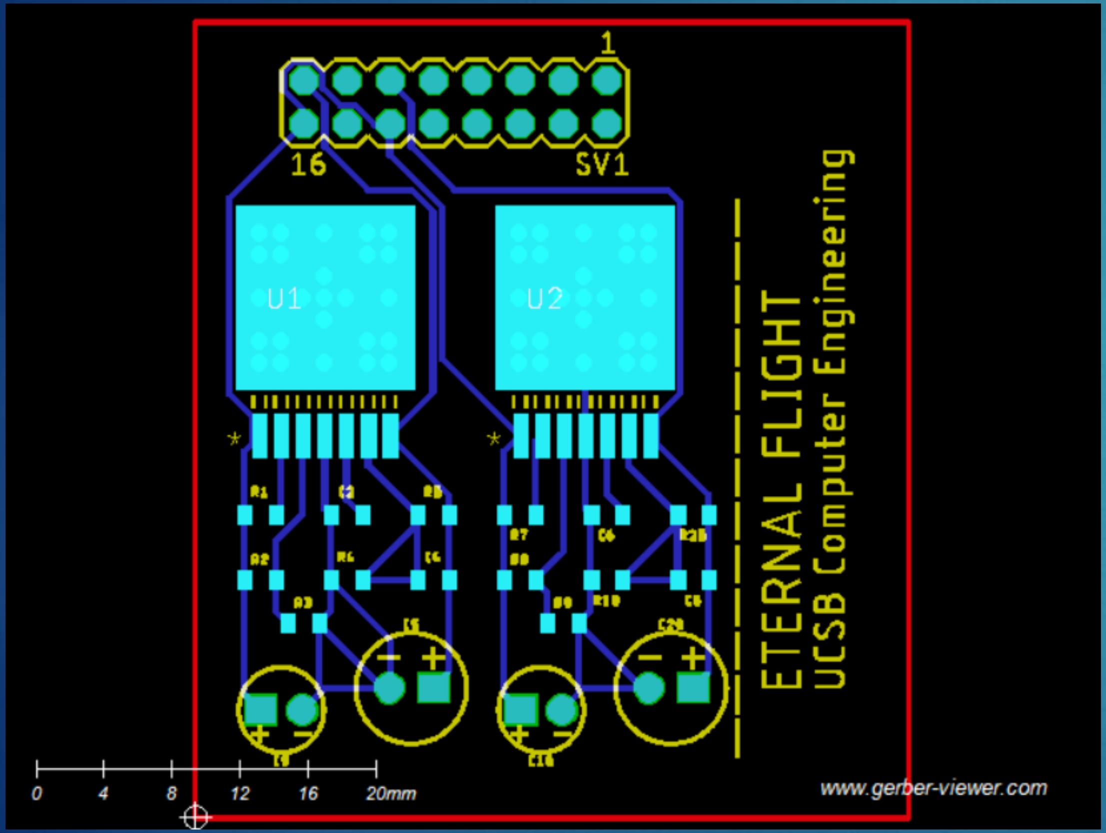
Final Design - Schematic

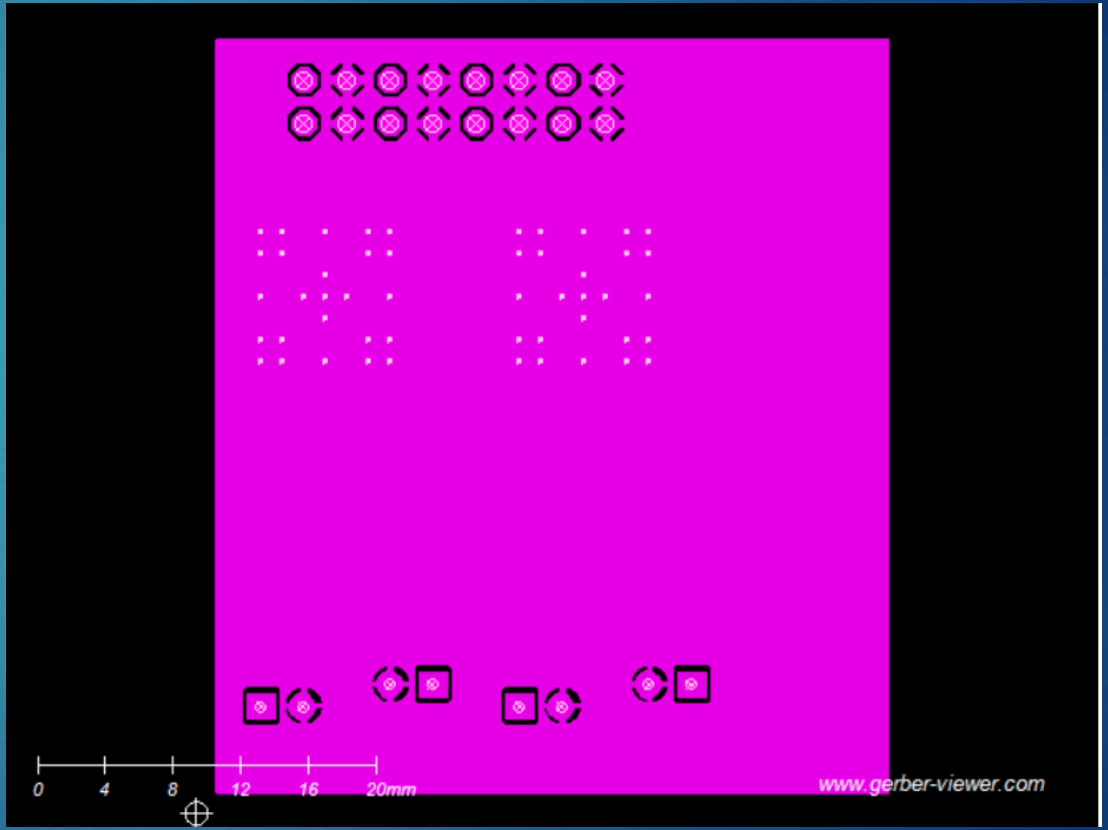


Final Design – PCB



Final Design – Gerber Files





General Guidelines

- Don't route directly under a chip unless you have to. You can route under in a different layer.
- If tying together multiple pins on a chip, bring the traces out rather than tying them directly together
- Make thermal pads slightly smaller
- Try to not make any signal wires travel a large distance or go through many vias
- 45 degree corners only
- Keep bypass capacitors close to the IC pins
- Ratsnest is a great tool for flooding planes
- Click on the + sign of the part to perform actions on it.
- Right click when routing to change the routing style. Leave it in the mode the outputs 45 degree angles.
- Modify the snapping of parts and routing using the grid menu. I recommend 10mil for the grid.

*Repeated here from homework