Intro to PCB Design

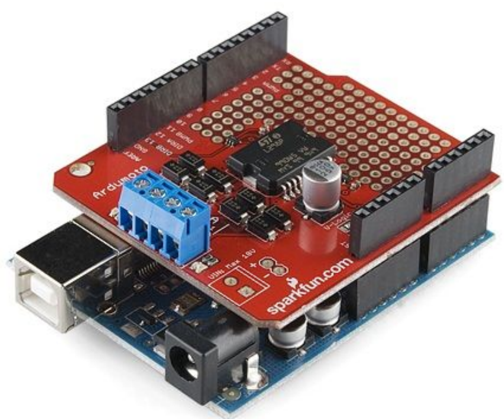
Objectives

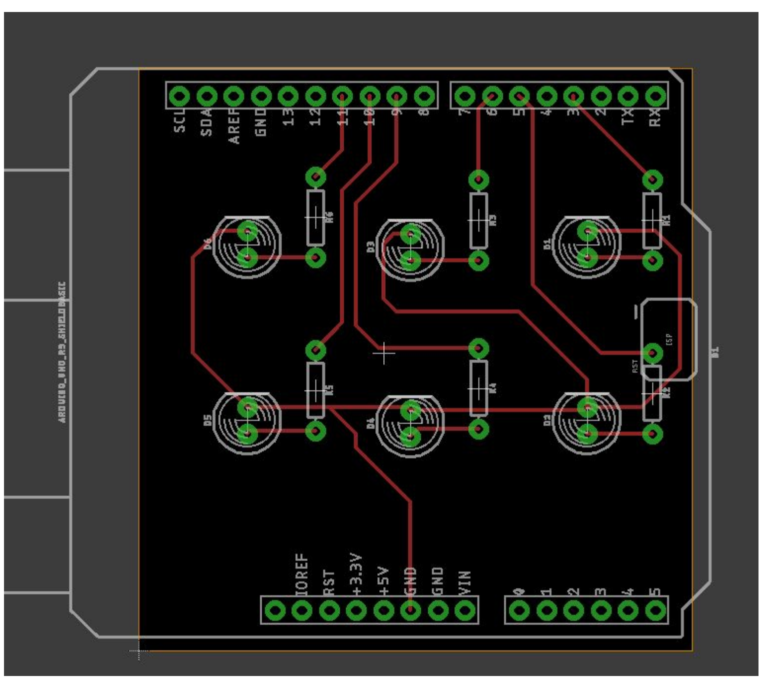
In this tutorial we'll design a printed circuit board (PCB) in Eagle. This PCB will be an Arduino shield - a board that sandwiches with an Arduino microcontroller to provide extra functionality. The PCB will have a collection of LEDs on it that can be controlled from the arduino.

- Get familiar with the process and vocabulary of PCB design
- Learn how to use Eagle
- Lay a foundation that you can build off of to do more complex PCB design in the future

You can learn more about Arduino shields here: https://learn.sparkfun.com/tutorials/arduino-shields/all

Questions? Email: katelyn.brinker@ieee.org or brinker@iastate.edu





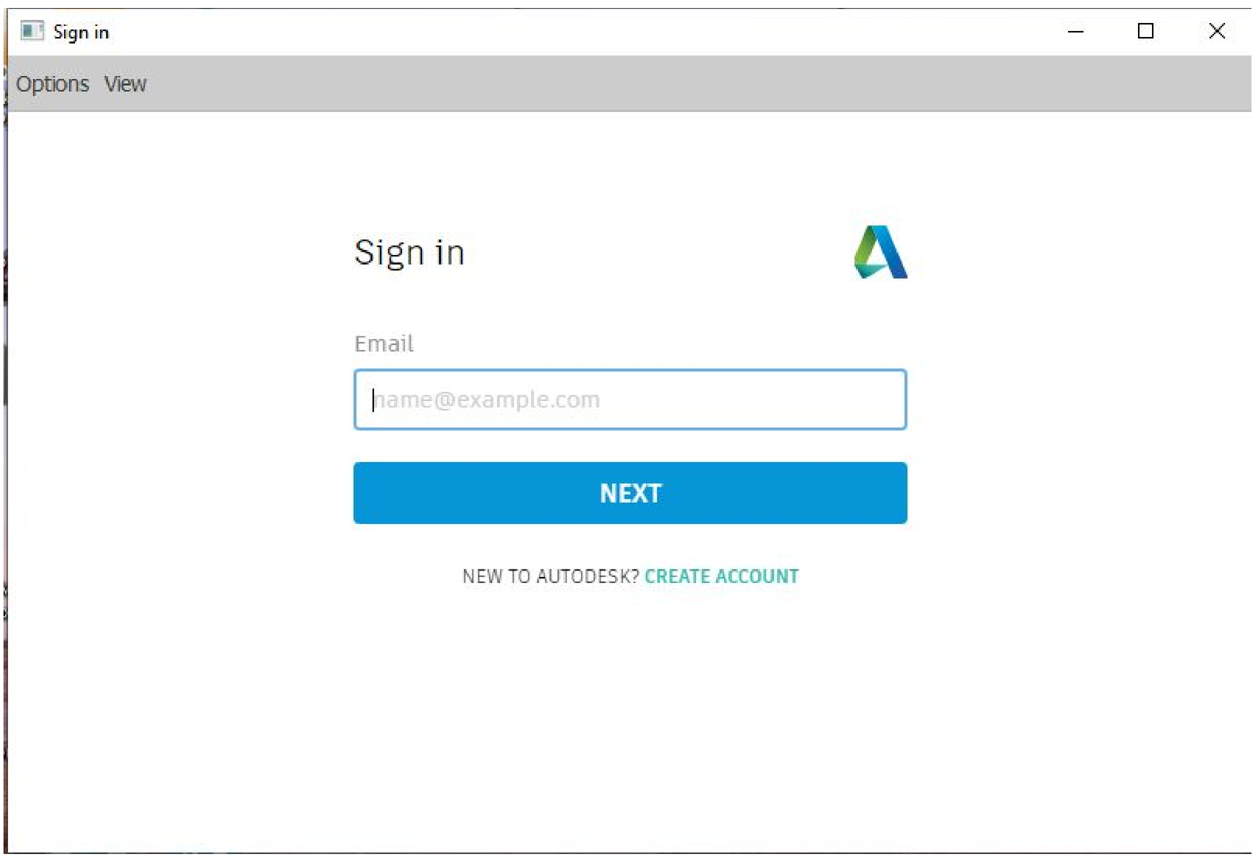
Getting Started

There are many different PCB design softwares, such as the following:

- Eagle
- Altium
- Cadence
- KiCAD
- CircuitMaker

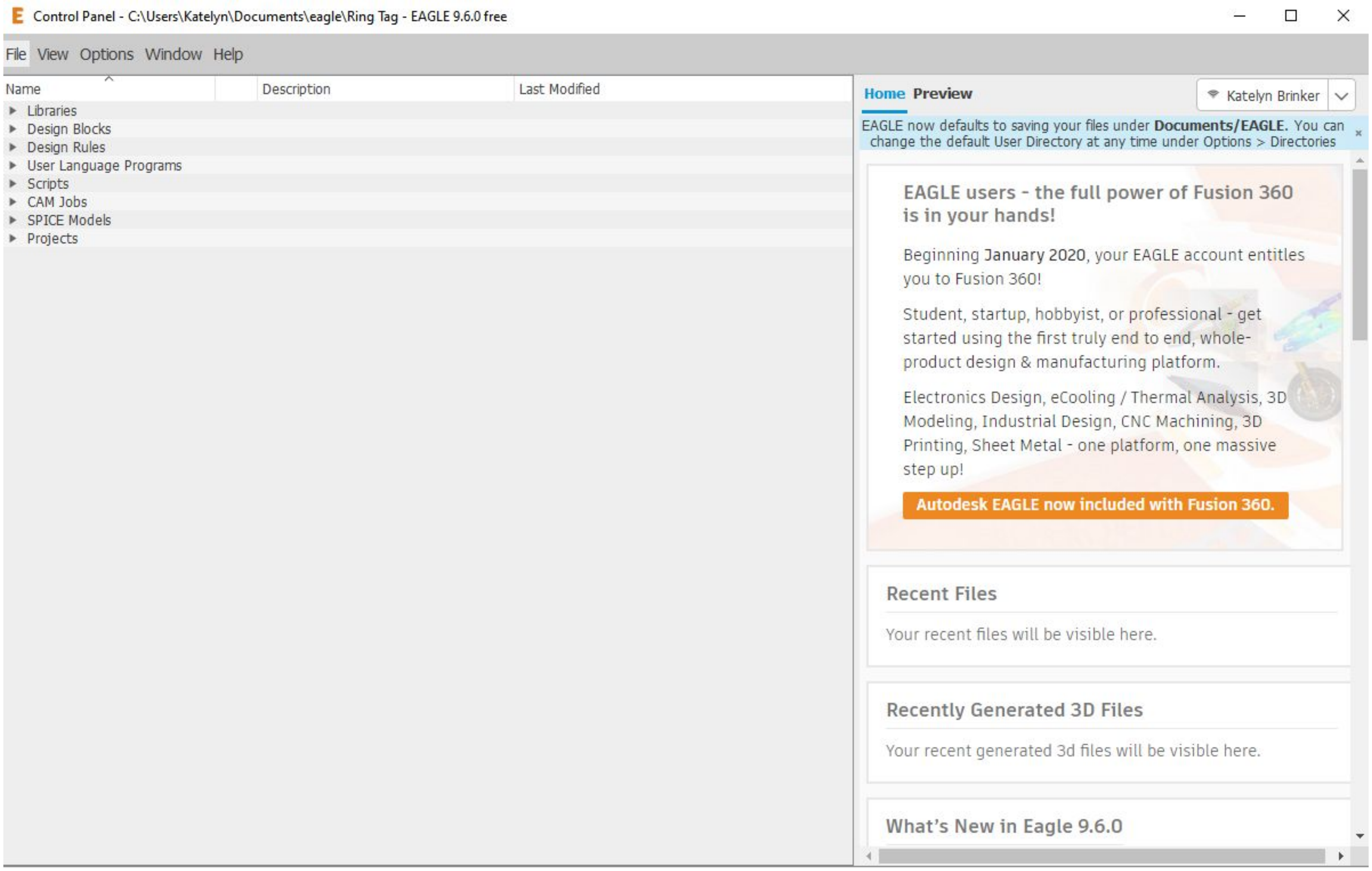
Eagle, KiCAD, and CircuitMaker have free versions. Altium and Cadence are both really powerful and customizable, but this makes them more difficult to learn on. The free version of Eagle is pretty user friendly and straightforward to learn on and provides the functionality we need for the workshop, so that's what we're going to use.

1. Download Eagle from the autodesk website: https://www.autodesk.com/products/eagle/free-download
2. After you download it, either create an Autodesk account or sign in with yours. After this is complete, Eagle will open in the Control Panel view.

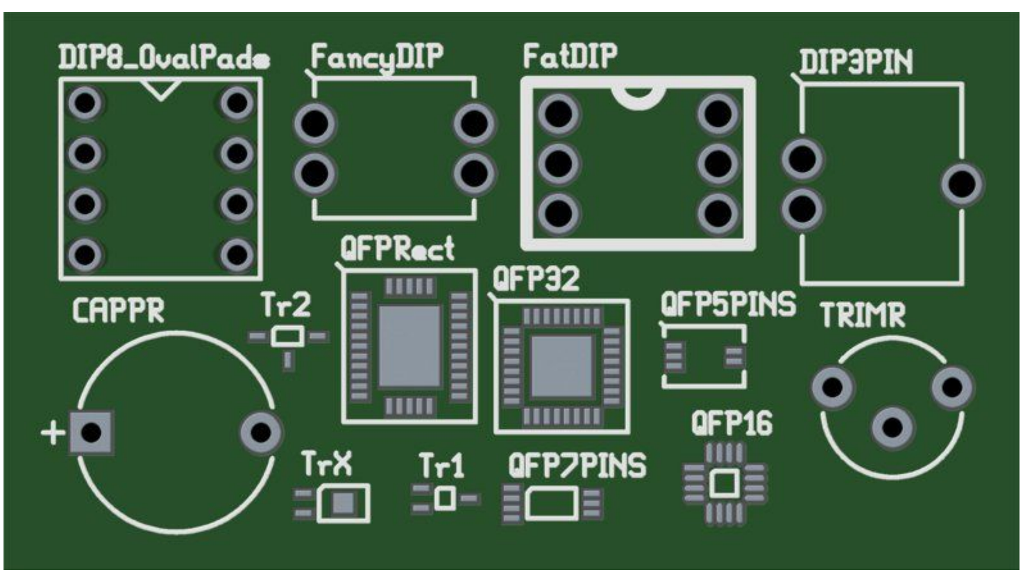


The Control Panel is your home base in Eagle. This is where you can create and access your projects and libraries.

Libraries are files that contain part descriptions: what is the symbol that represents the part and what does the part's physical footprint need to look like.

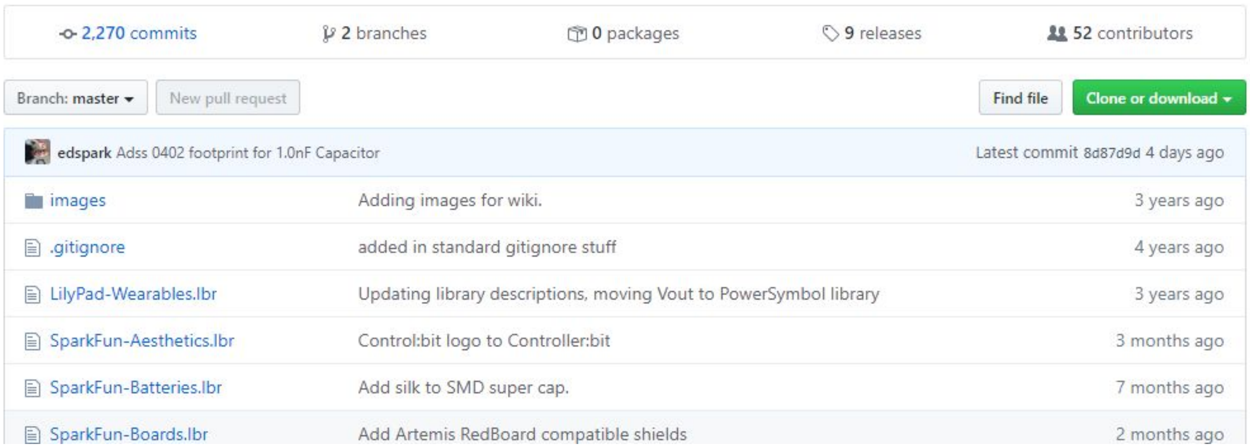


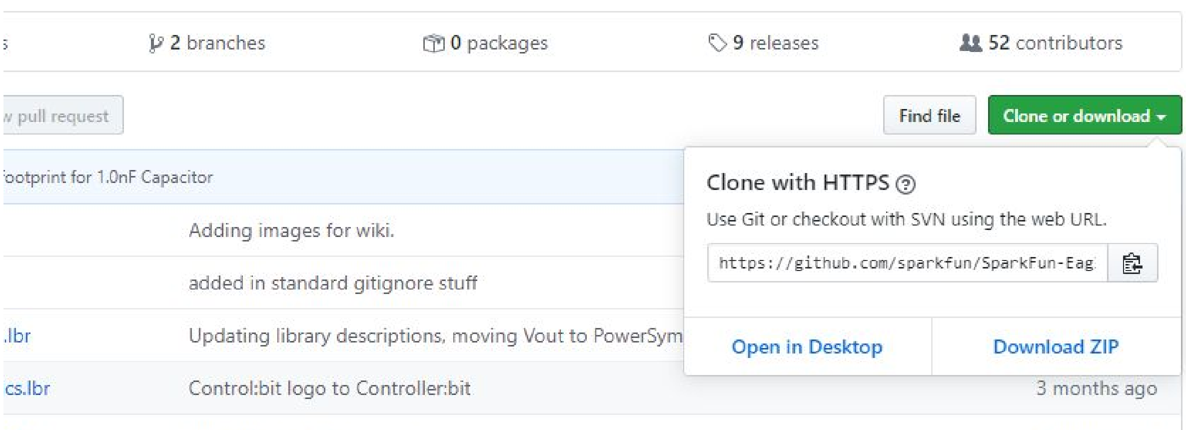
The footprint is the layout of the pads that the component will be soldered to.



You can make your own libraries or you can download premade libraries. We're going to be using the SparkFun Eagle libraries that are on github.

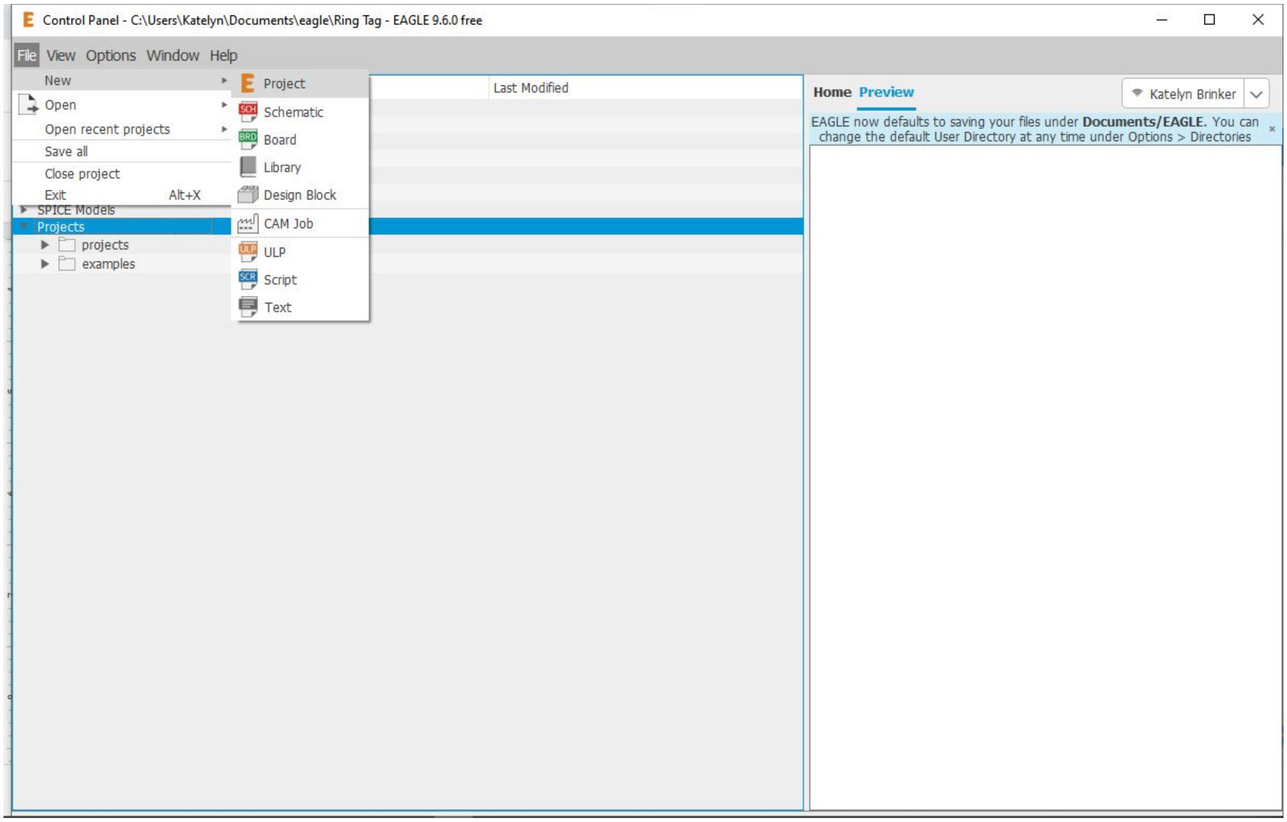
3. Download the SparkFun Eagle libraries from github using this link: https://github.com/sparkfun/SparkFun-Eagle-Libraries Click on "Clone or download" and then select "Download ZIP"
4. Extract the library files from the zip file.

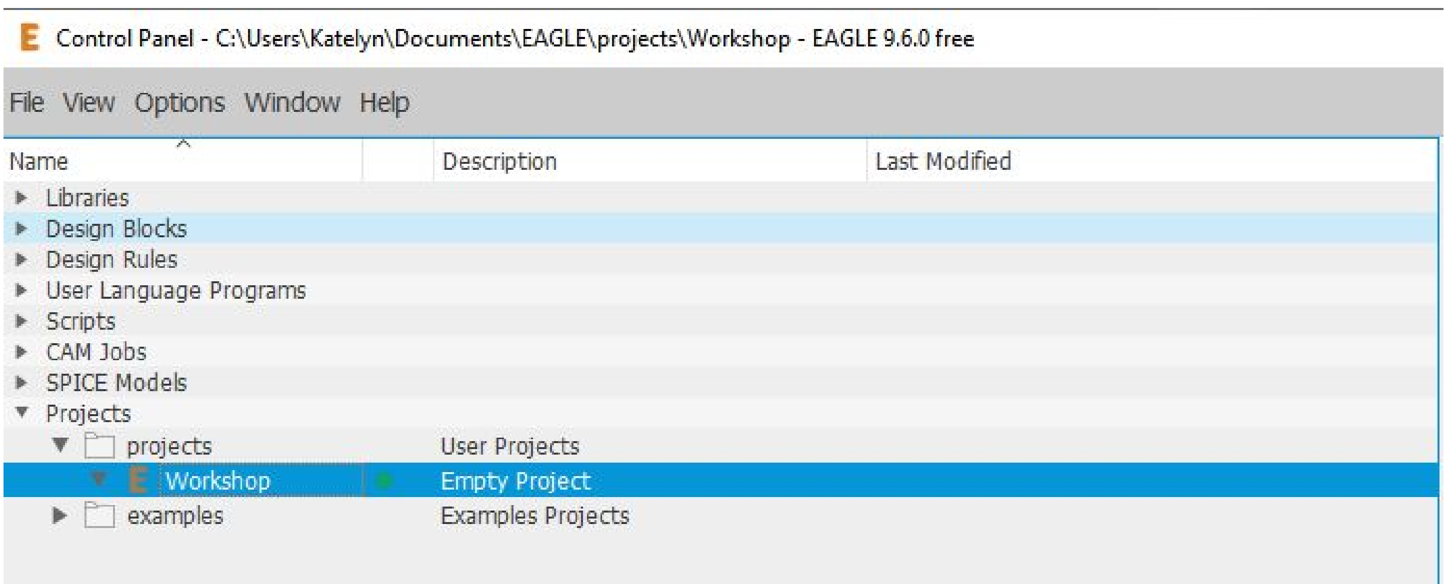




We'll include the libraries once we start laying out the schematic. But first, we need to start a "Project" for our board to reside in.

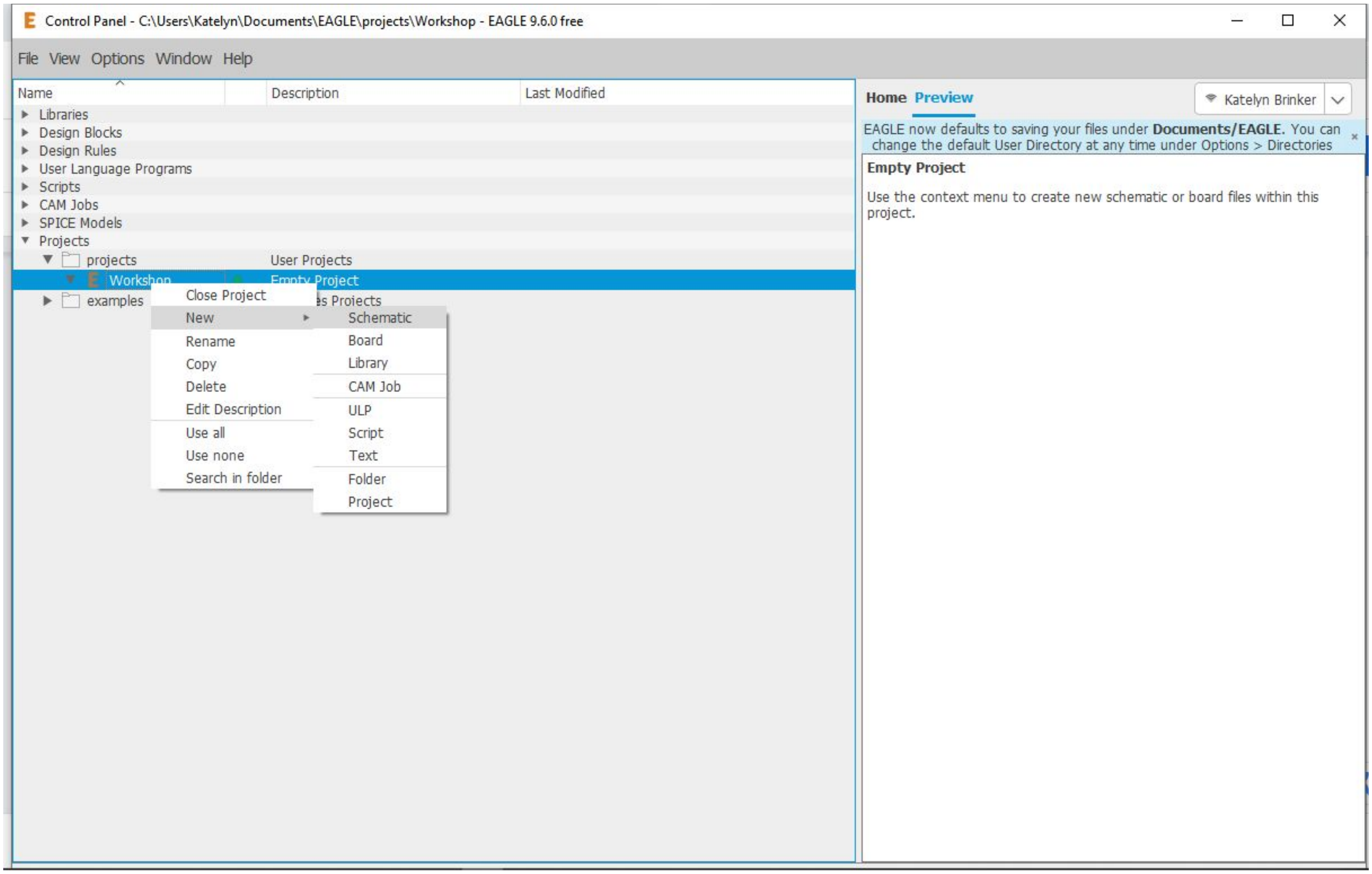
5. In the top menu of the Control Panel select File - New - Project. This will add a new project under the "projects" tab in the Control Panel. Rename the new project whatever you want. In this tutorial it is named "Workshop." The project description for the project you just created should be "Empty Project"

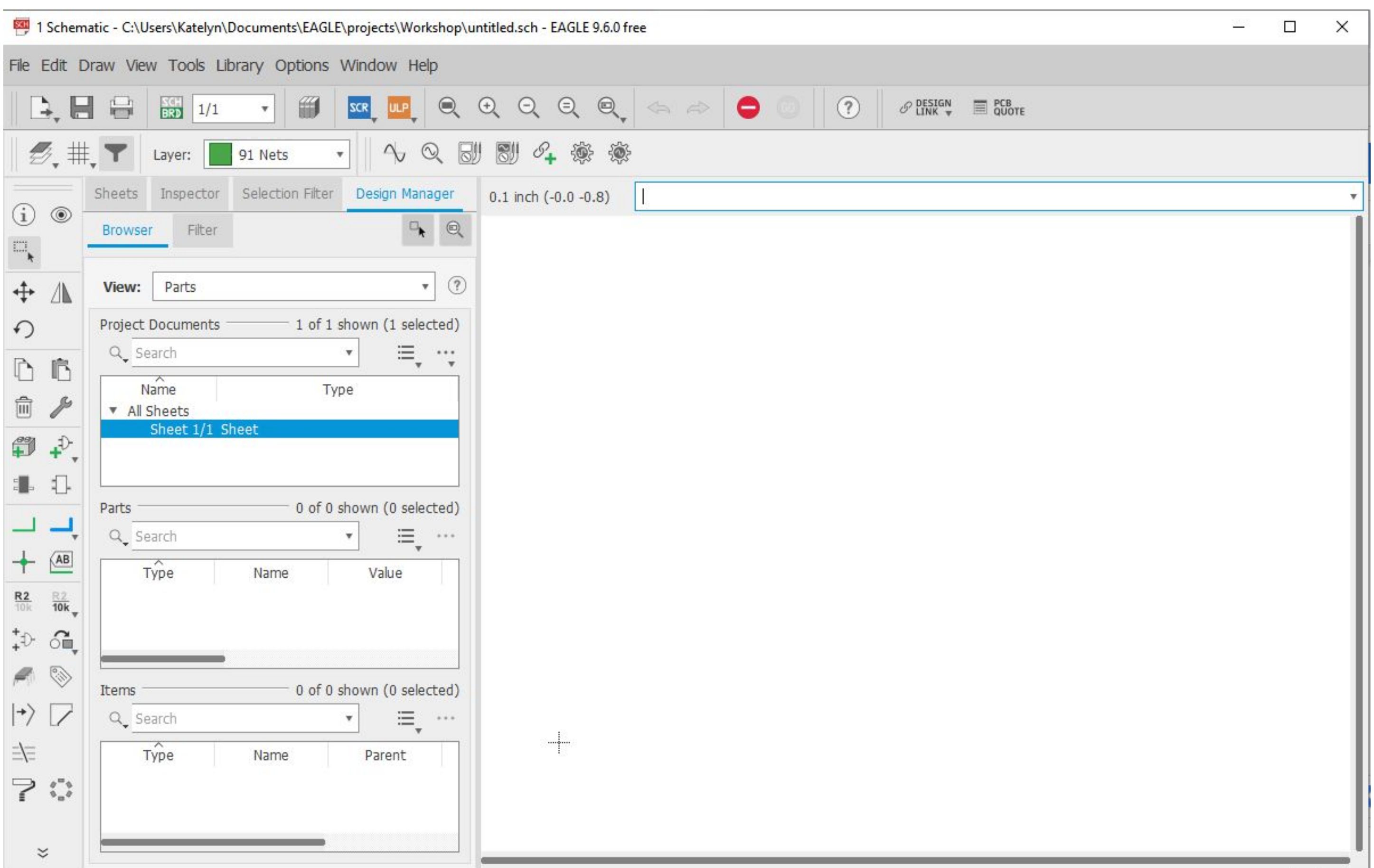




We'll add the schematic and layout to the project so that all the pieces we need for making a PCB are grouped together.

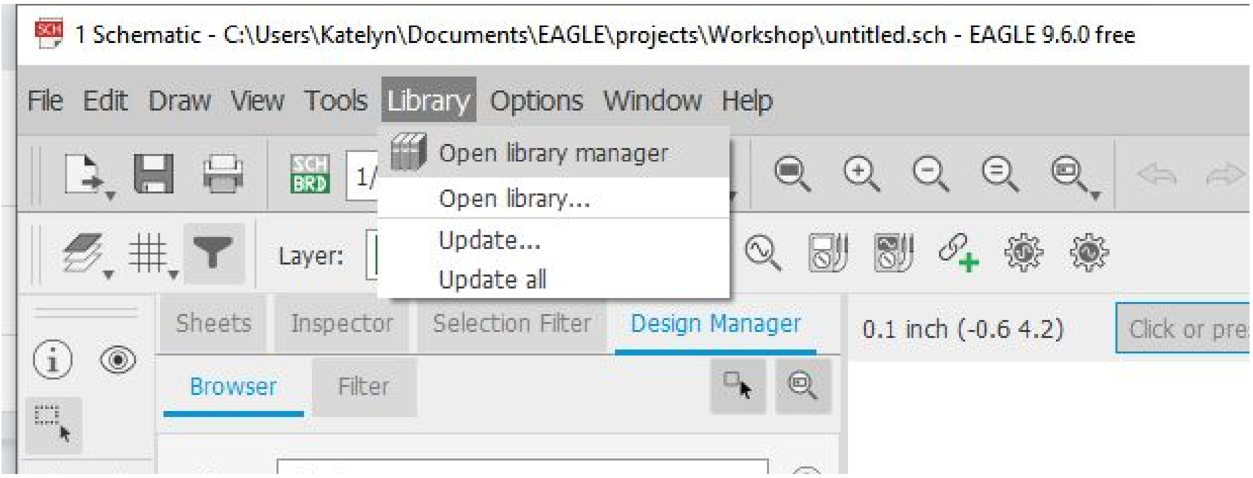
6. Right click on your project in the Control panel then select New - Schematic. A schematic will then be added to your project and will open in a new window.


Control Panel - C:\Users\Katelyn\Documents\EAGLE\projects\Workshop - EAGLE 9.6.0 free
File View Options Window Help
Name
Description
Last Modified
Libraries
Design Blocks
Design Rules
User Language Programs
Scripts
CAM Jobs
SPICE Models
Projects
projects
User Projects
Workshop
Empty Project
examples
Close Project
New
Rename
Copy
Delete
Edit Description
Use all
Use none
Search in folder
Schematic
Board
Library
CAM Job
ULP
Script
Text
Folder
Project
Home Preview
Katelyn Brinker
EAGLE now defaults to saving your files under Documents/EAGLE. You can change the default User Directory at any time under Options > Directories
Empty Project
Use the context menu to create new schematic or board files within this project.

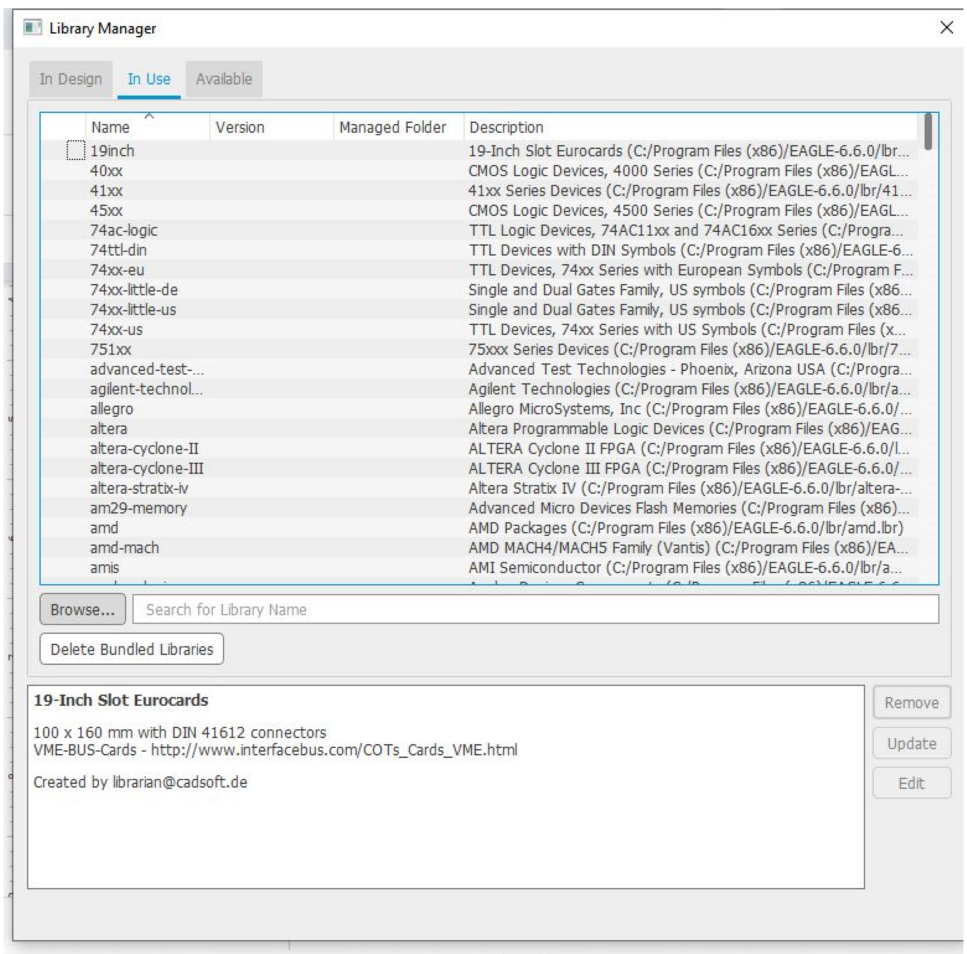

1 Schematic - C:\Users\Katelyn\Documents\EAGLE\projects\Workshop\untitled.sch - EAGLE 9.6.0 free
File Edit Draw View Tools Library Options Window Help
1/1
SCR
ULP
DESIGN LINK
PCB QUOTE
Layer:
91 Nets
Sheets
Inspector
Selection Filter
Design Manager
0.1 inch (-0.0 -0.8)
Browser
Filter
View:
Parts
Project Documents
1 of 1 shown (1 selected)
Search
Name
Type
All Sheets
Sheet 1/1 Sheet
Parts
0 of 0 shown (0 selected)
Search
Type
Name
Value
Items
0 of 0 shown (0 selected)
Search
Type
Name
Parent

We're going to add the sparkfun libraries through the library manager before we layout the schematic.

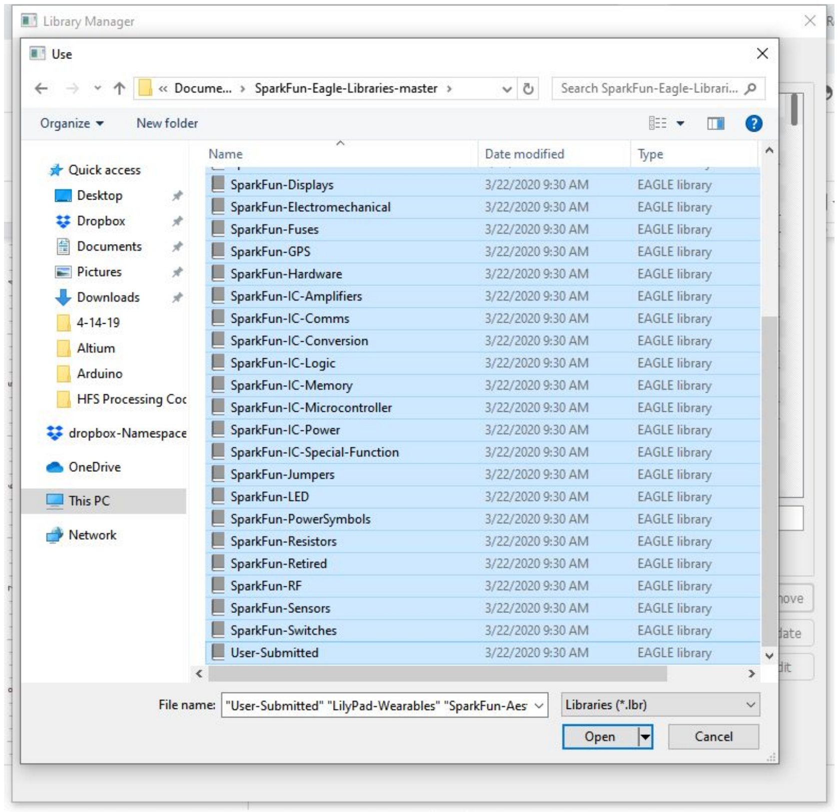
7. From the top menu select "Library" and then click on "Open library manager"



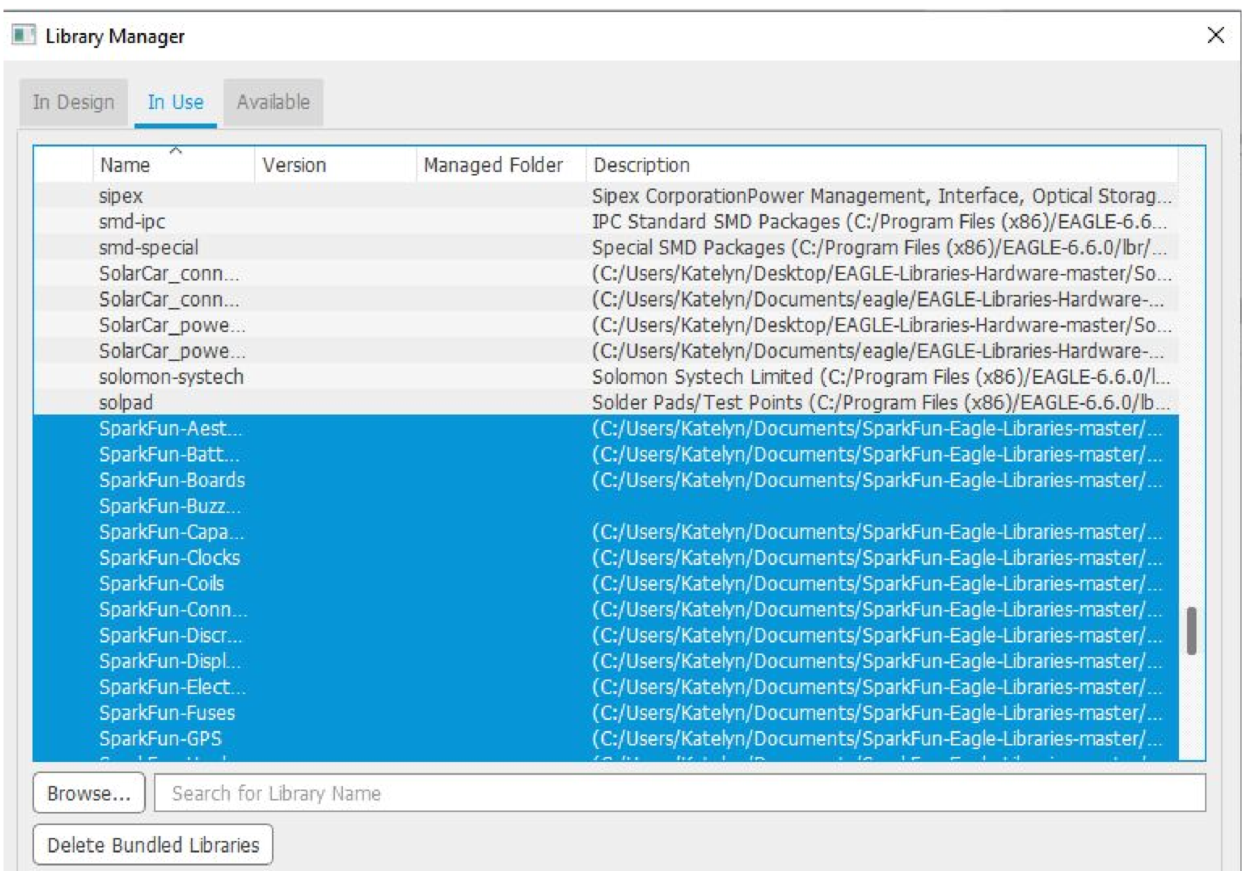
8. Navigate to the "In Use" tab of the Library Manager and then hit the "Browse" button.



9. In the pop up window, browse to where you extracted the SparkFun library files previously. Select all the individual library files and then hit the Open button.



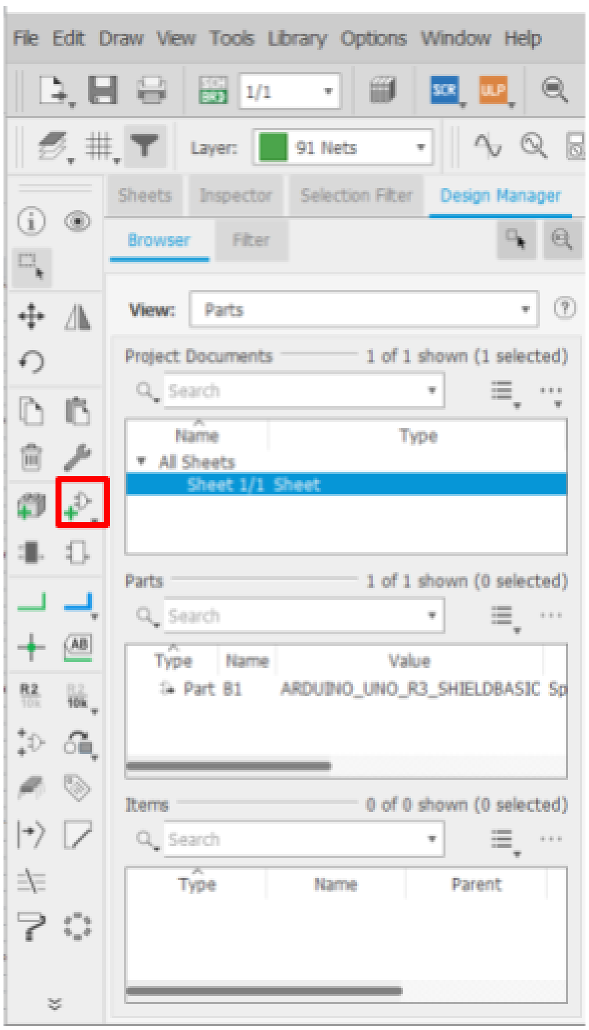
10. In the library manager under the “In Use” tab, scroll down and make sure the SparkFun libraries appear in the list.



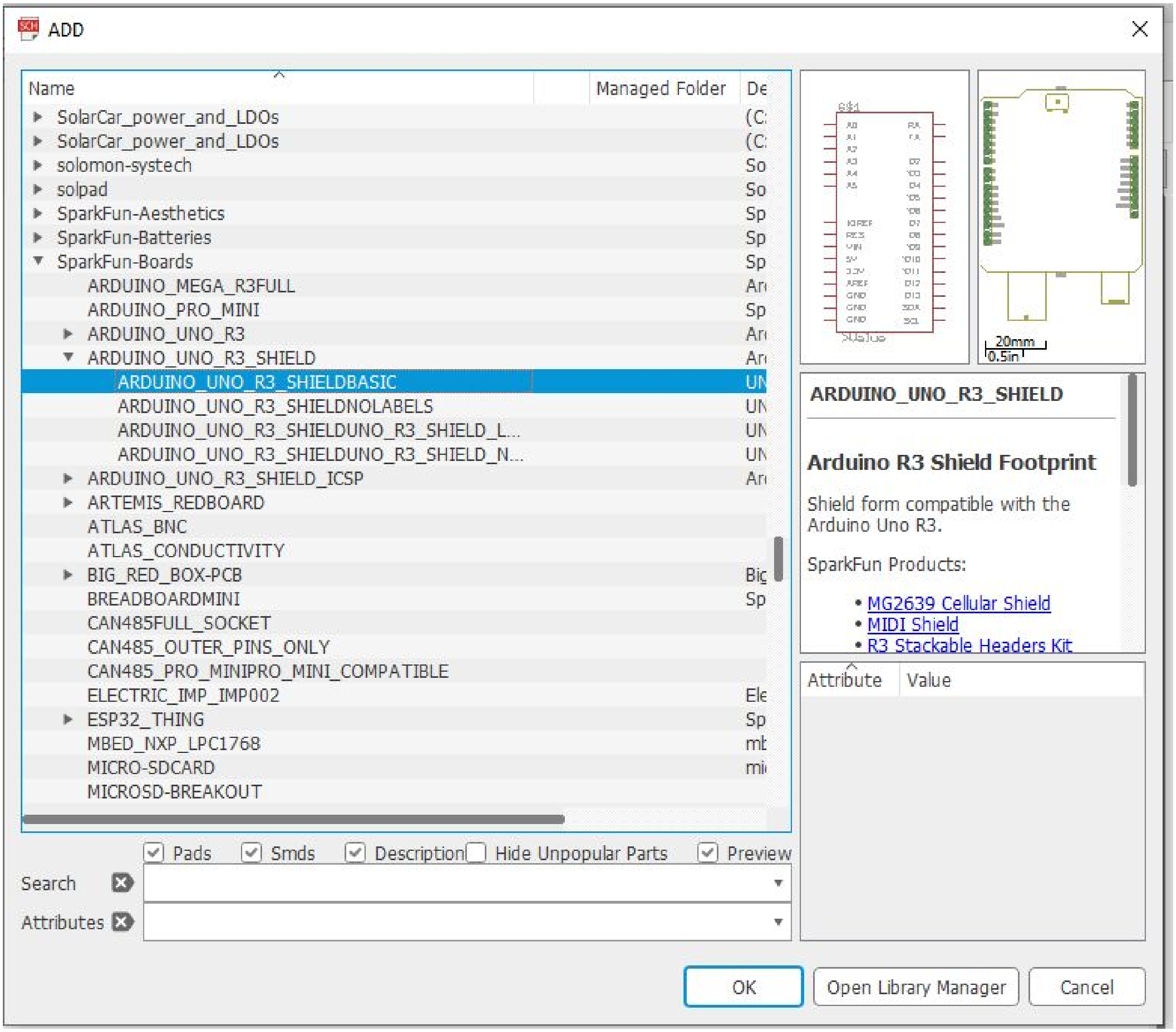
Schematic Layout

The schematic is the symbolic description of the PCB. It's analogous to a circuit diagram. It's where we'll indicate the connections between parts and it'll help us when we do the board layout later.

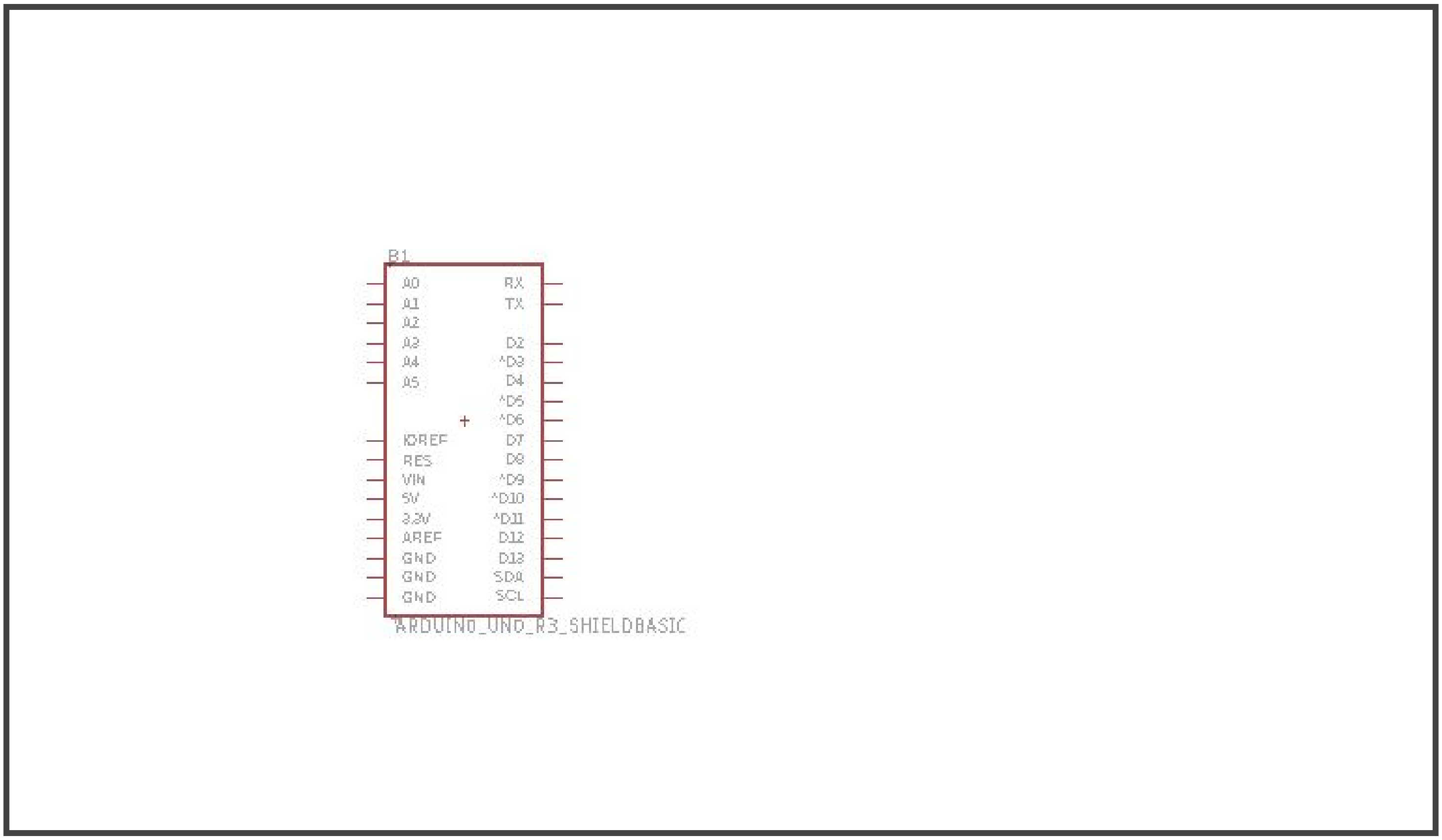
We'll use the tools in the toolbar on the left to build the schematic. The button to add components is boxed in red in the image below. Hovering over this tool brings up the description "Add Part"



11. Save your schematic by hitting the save button in the upper left or through File - Save
12. Click on the Add Part tool. In the Add window that pops up, navigate to SparkFun-Boards, then ARDUINO_UNO_R3_SHIELD, and then select ARDUINO_UNO_R3_SHIELDBASIC. A preview of the schematic symbol and footprint will show in the Add window. Select OK.



13. Place an instance of the Arduino Uno shield symbol in the schematic window by clicking once in the white canvas in the schematic window. Clicking again will place another instance of the symbol. Press the escape button on your keyboard to exit the symbol placement mode. This will bring you back to the Add window. Pressing escape again will close the add window.



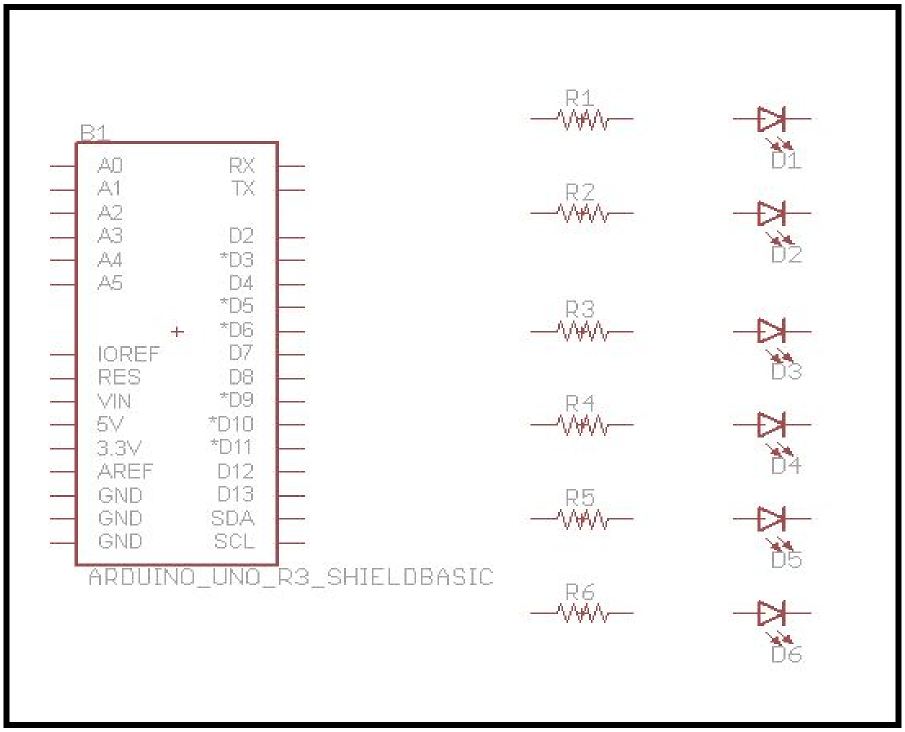
To pan the schematic canvas, hold down your mouse's scroll wheel and drag the canvas. To zoom in, roll the scroll wheel away from you. To zoom out, roll the scroll wheel towards you.

You can rotate a symbol by right clicking before you left click to place it in the schematic.You can also right click on the symbol and then select rotate.

14. Place the components in the BOM below in the schematic. The Library column tells you where to find the part in the Add window. Remember that you can right click before placing a component to rotate it. After placing components, you can use the move tool in the left hand toolbar to arrange the symbols nicely.

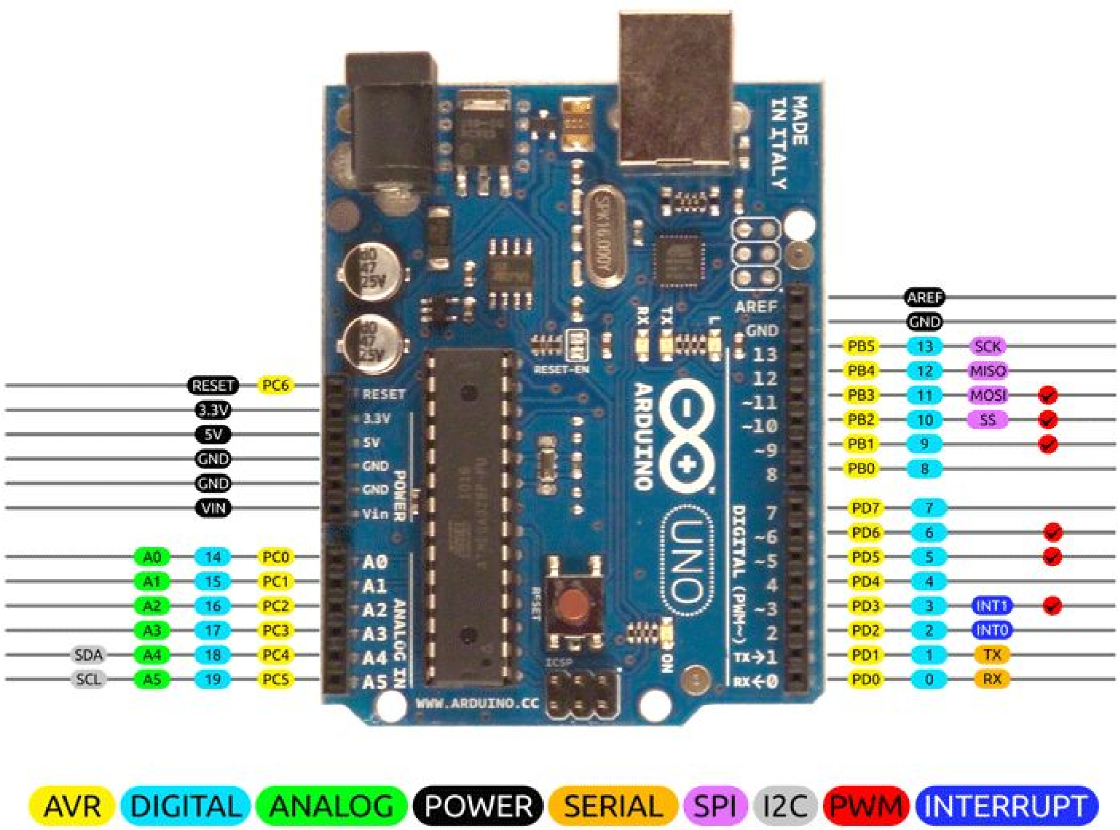
Bill of Materials (BOM)

Part Name	Library/Folder	Quantity
ARDUINO_UNO_R3_SHEILDBASIC	SparkFun-Boards -> ARDUINO_UNO_R3_SHIELD	1
RESISTORAXIAL-0.3	Sparkfun-Resistors ->RESISTOR	6
LED5MM	Sparkfun-LED ->LED	6



The resistors will be used to limit the current going into the LEDs. We want to connect each LED to a resistor and to a digital pin on the Arduino through the shield board. The full pinout of the Arduino is shown below. All the pins whose names start with “D” (e.g., D2 or D7) are digital pins.

Arduino Pinout

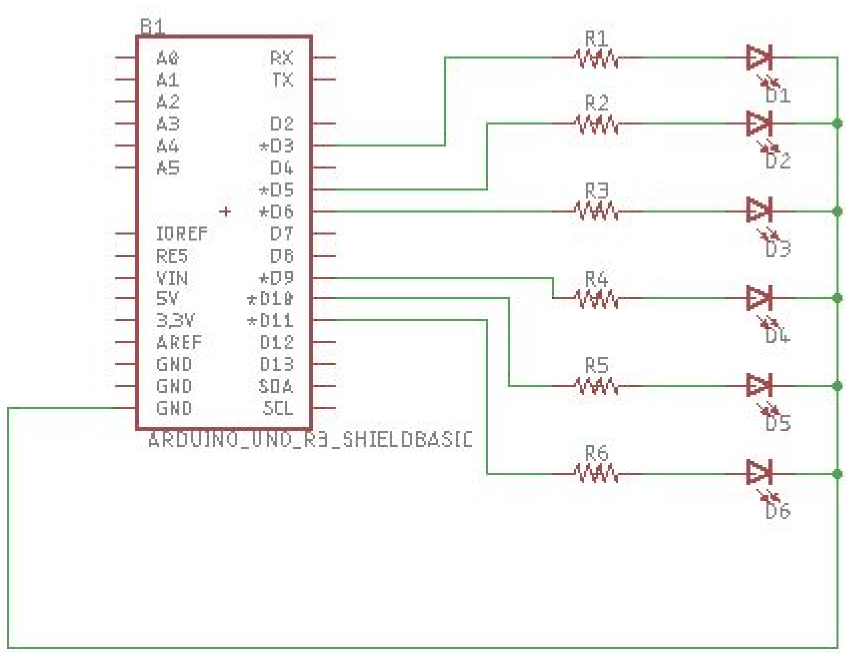


15. Use the Net tool (green L shape button) in the toolbar to connect the symbols according to the suggested connections and suggested schematic below. To use the Net tool, click on the pin of a symbol to start the net and then click on the pin of the component you want to connect to.
16. Save your schematic.

Suggested Board Connections

Arduino Shield Pin	Component Connection
D3	R1
D5	R2
D6	R3
D9	R4
D10	R5
D11	R6
GND	All LED cathodes

Suggested Schematic

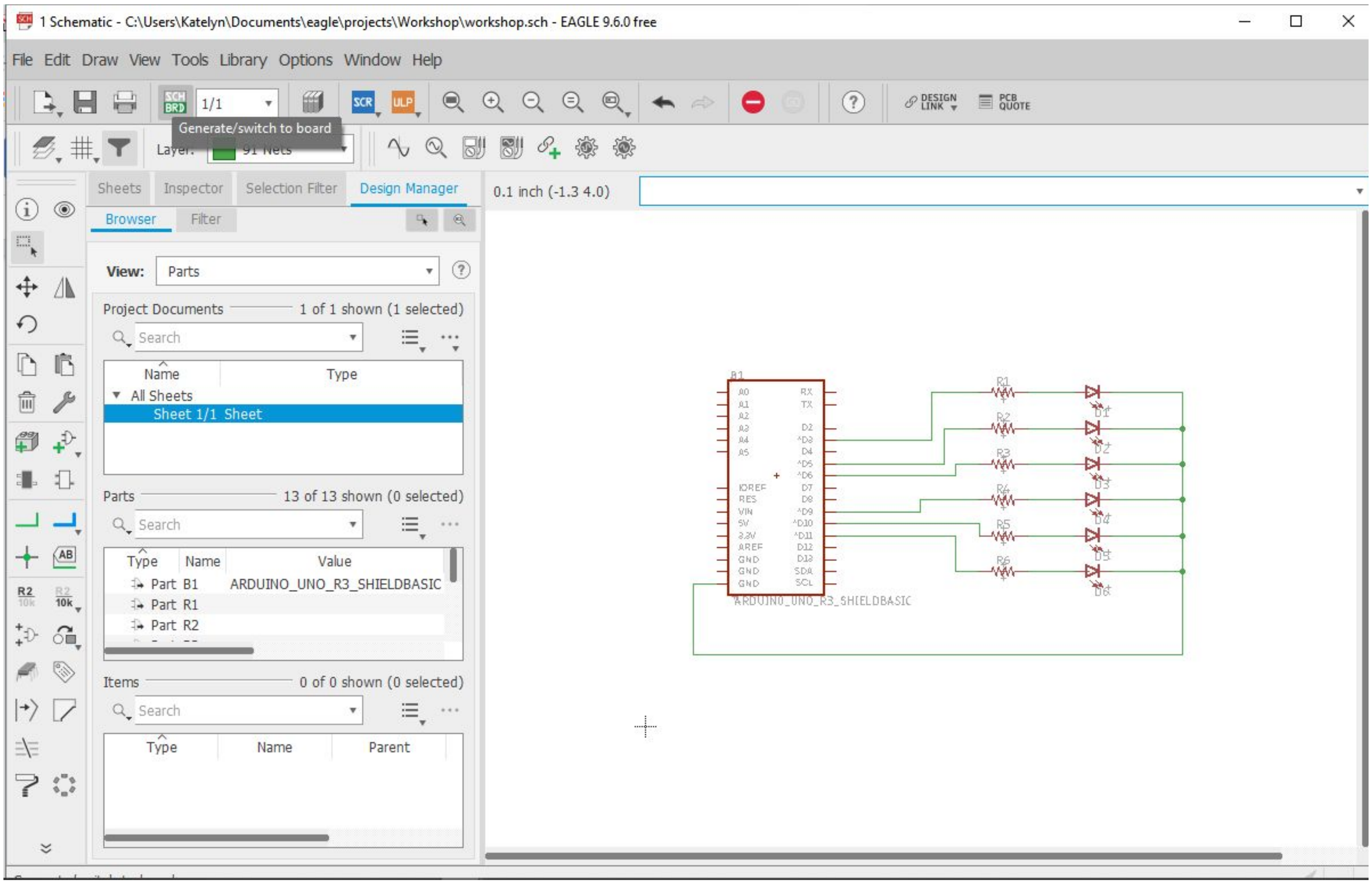


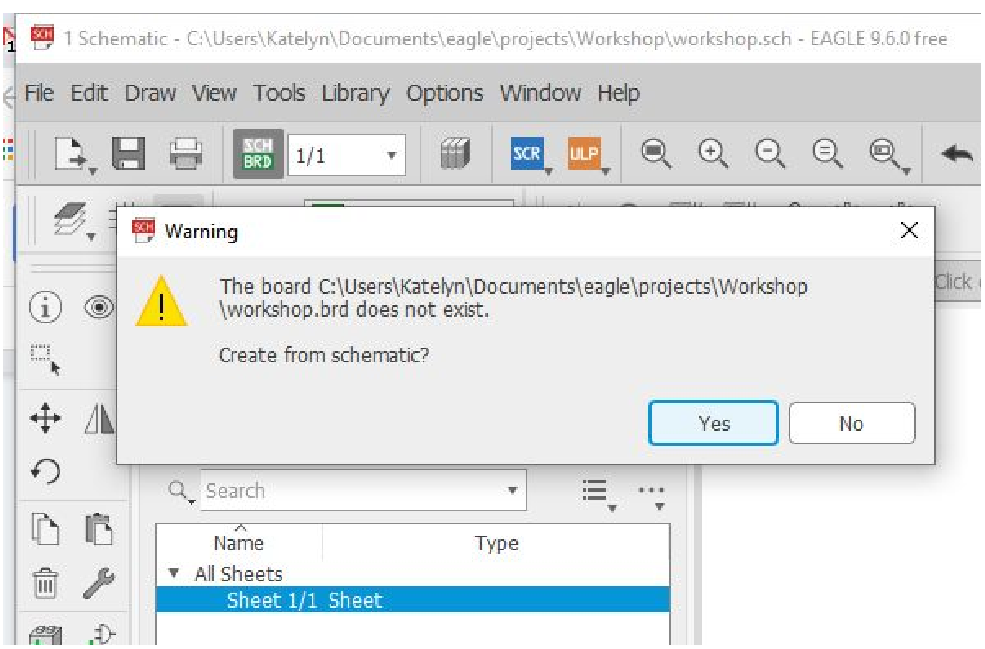
NOTE: There are many possible ways to layout the schematic (i.e., different pin connections and organizations of symbols). This schematic layout will make laying out the board easy and is therefore suggested.

Board Layout

Now that the schematic is complete, we can start the board layout.

17. To create the board, press the “Generate/switch to board” button in the top menu. Select “Yes” in the pop up warning to create the board from the schematic. A new Board window will open.

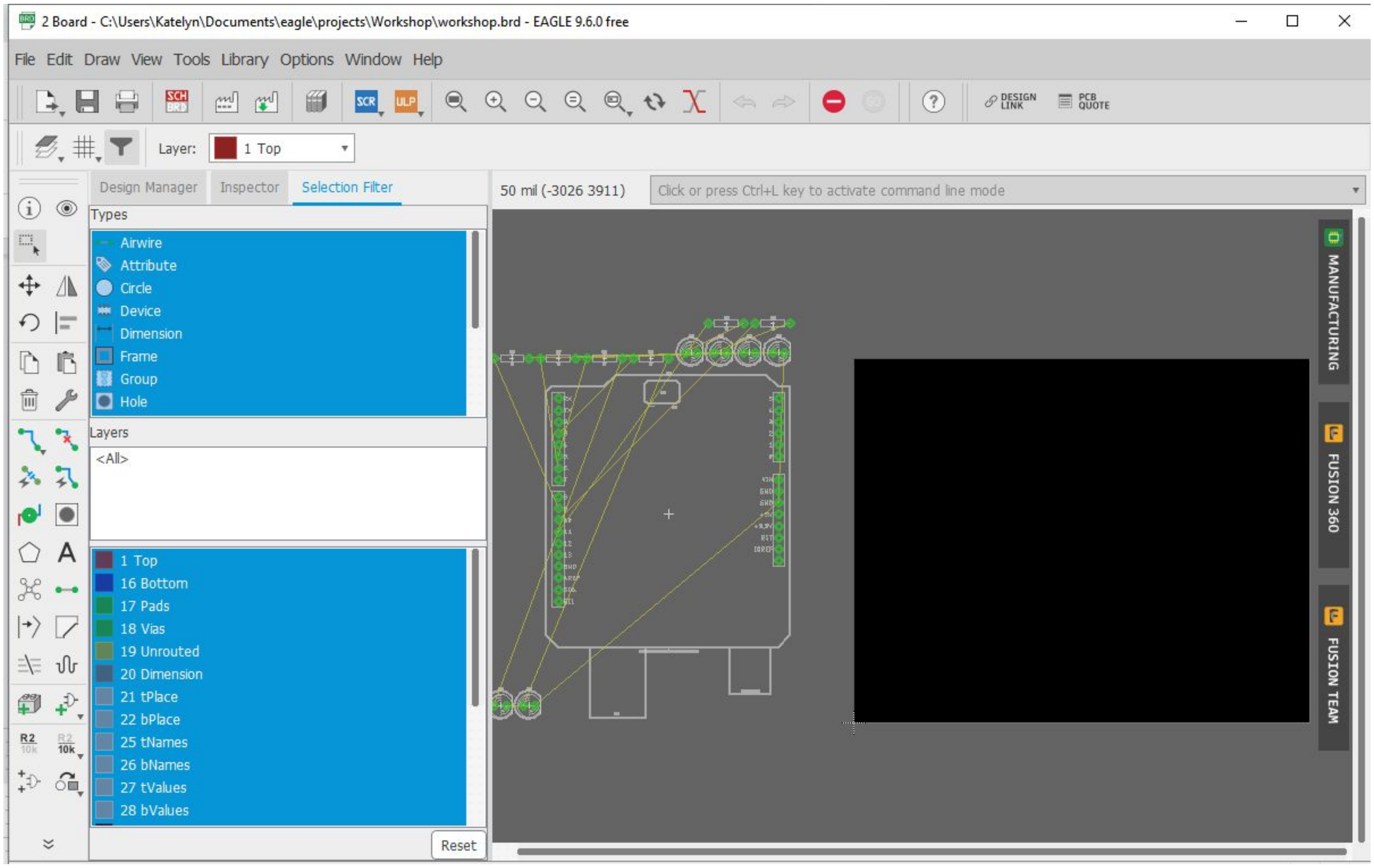




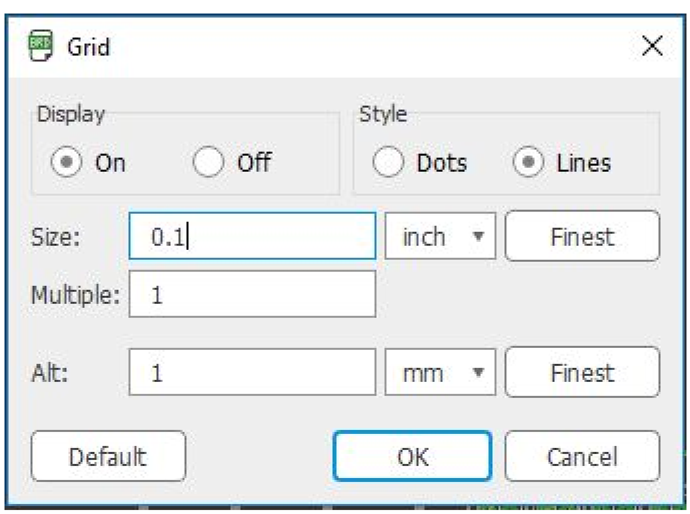
The board window will show the footprints of all the components you had in your schematic. The green circles represent copper pads where through hole components will go in the physical board.

The yellow lines connecting components are called airwires and they represent the connections we made in the schematic. In laying out the board, we will replace the airwires with traces.

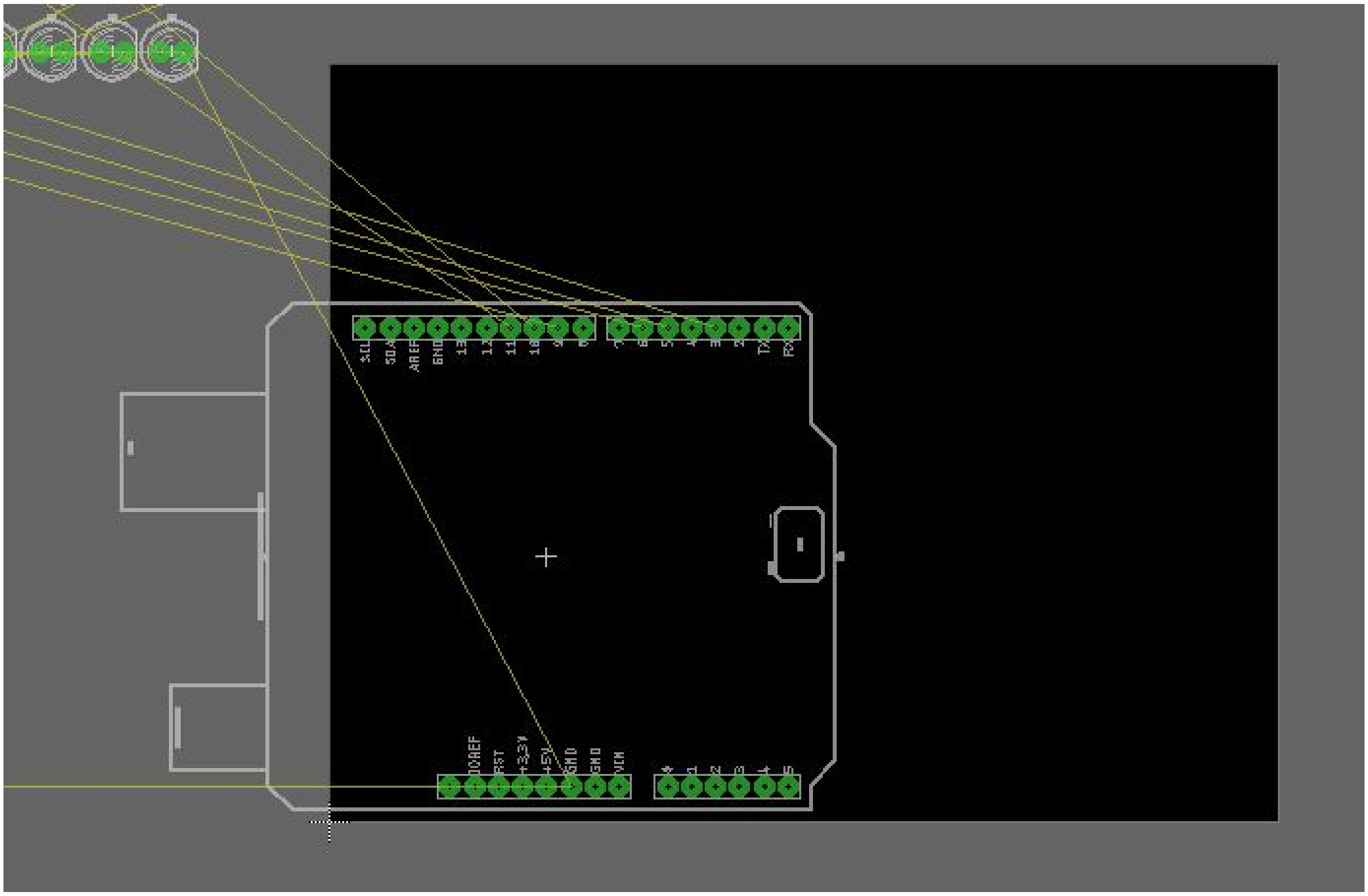
The black area in the board window is the board area. We can adjust the size and shape of the board area based on what we want to fabricate.



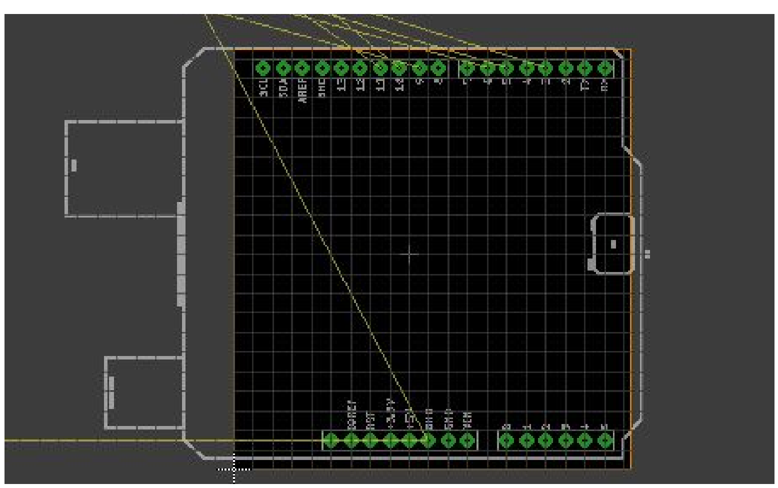
18. Adjust your grid - this is a personal preference.The grid will determine how you can move components. You can choose whether or not to display it.



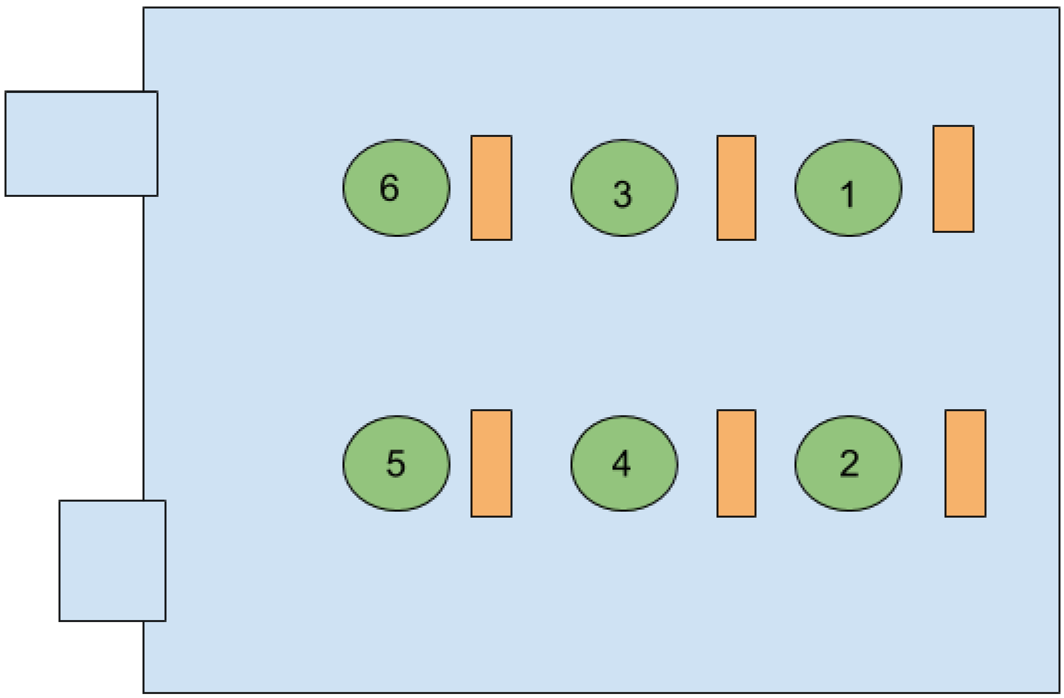
19. Move the Arduino shield into the board area using the move tool. Once you select the shield with the move tool (click on the plus in the middle of the shield outline) you can right click to rotate the shield before placing it. We're going to let the USB and ethernet ports hang off the edge. Since this will be stacking on top of an arduino, we just need to make sure all of our header pins will be in the board profile.

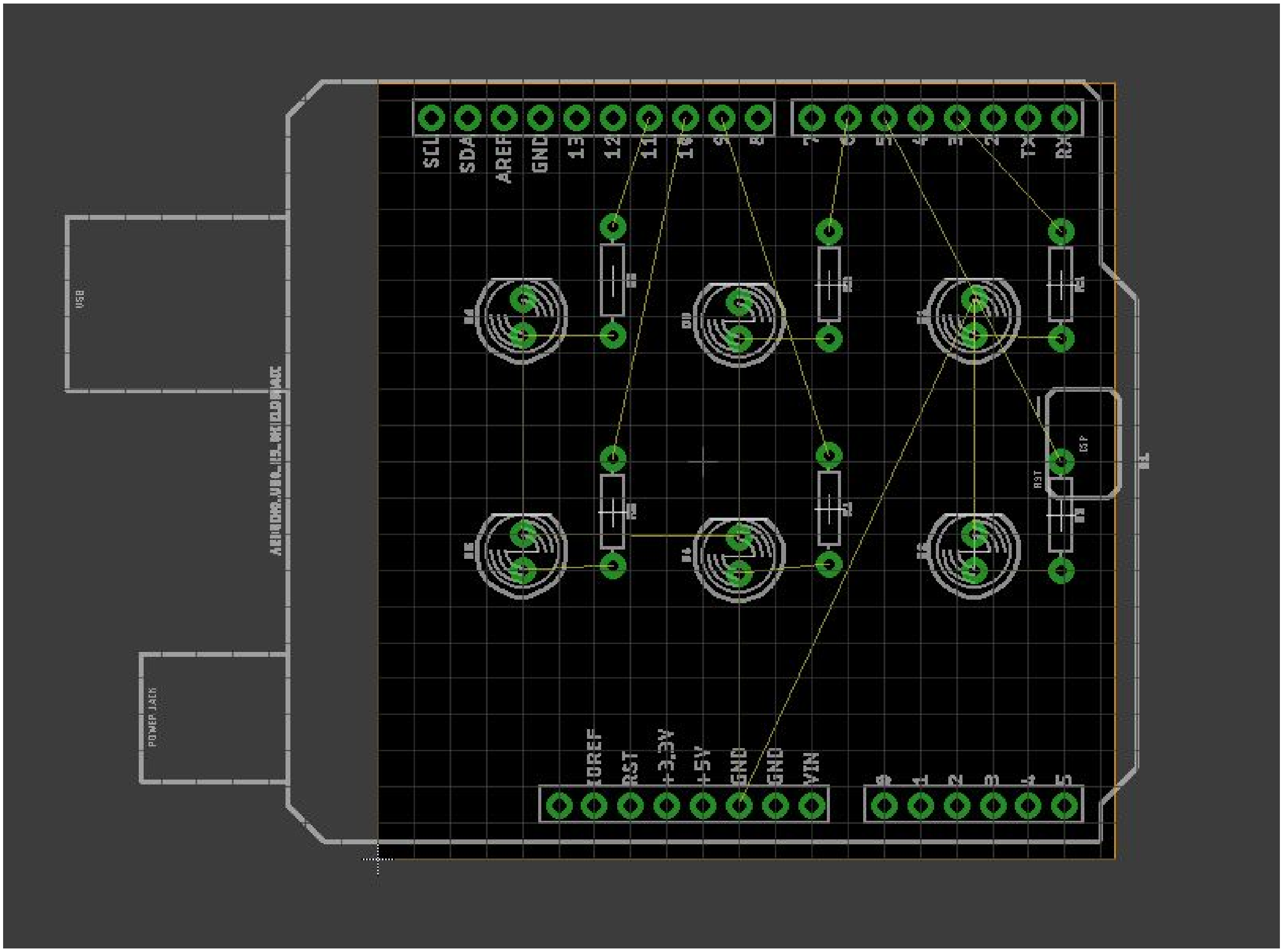


20. Adjust the board size using the move tool so it fits the profile of the arduino. With the move tool selected, click on an edge of the board and drag it in towards the shield. You can also right click on an edge of the board and select properties. In the properties window you can adjust the position of the edge numerically.



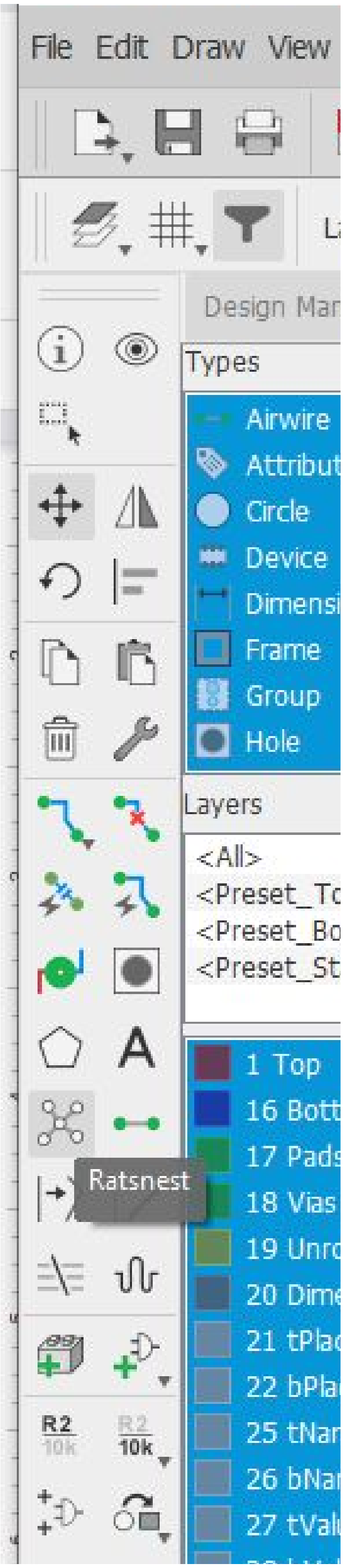
21. Place your remaining components inside your board area using the move tool following the suggested layout. Zoom in to see the component designations (e.g., D3 or R1). It's ultimately up to you how you want to place your components but you want to be able to make connections where all the airwires are without crossing connections and you want your connections to be as direct as possible.
    A. Right click to rotate a component
    B. Use the mouse scroll wheel to zoom in and out
    C. Hold down the scroll wheel to pan around your layout
    D. Hold down the alt key while you move components to move finer increments than your normal grid





The Ratsnest button can be used to clean up airwires. It'll find the most direct connections and adjust for you. The Ratsnest button can be found in the toolbar on the left.

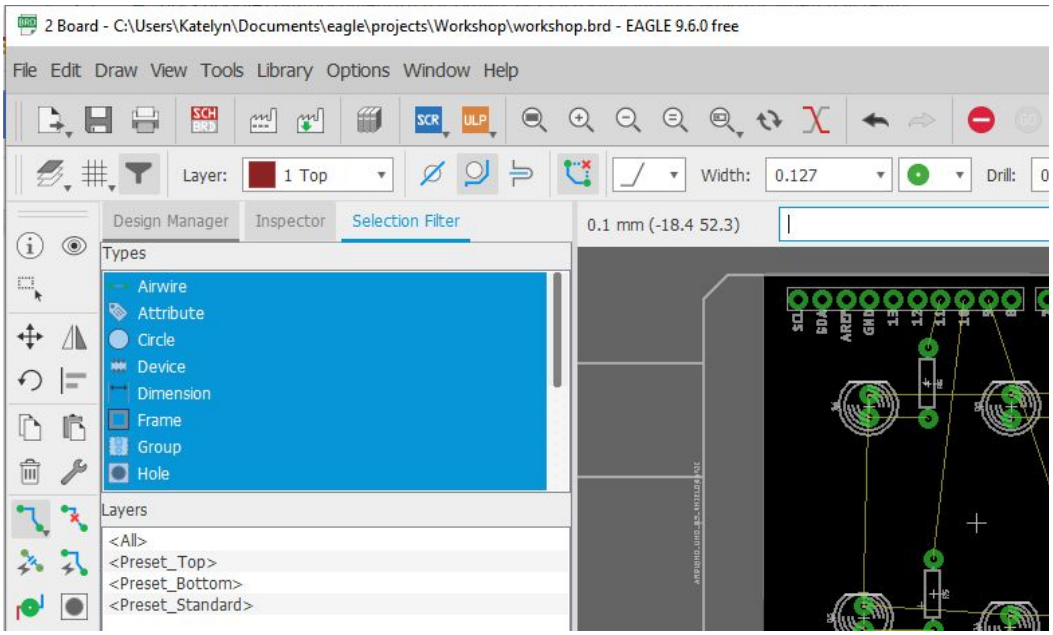
22. Click the Ratsnest button after placing your components in the board area.



Next, we're going to route the board. Routing is the process of placing traces where the airwires are.

The Route Airwire button is located in the left hand toolbar above the Ratsnest button.

23. Select the route button. In the top of the window, adjust the trace settings so that the width is 0.254, the top layer is selected, the walkaround obstacles setting is selected, and the 45 degree bend option is selected.



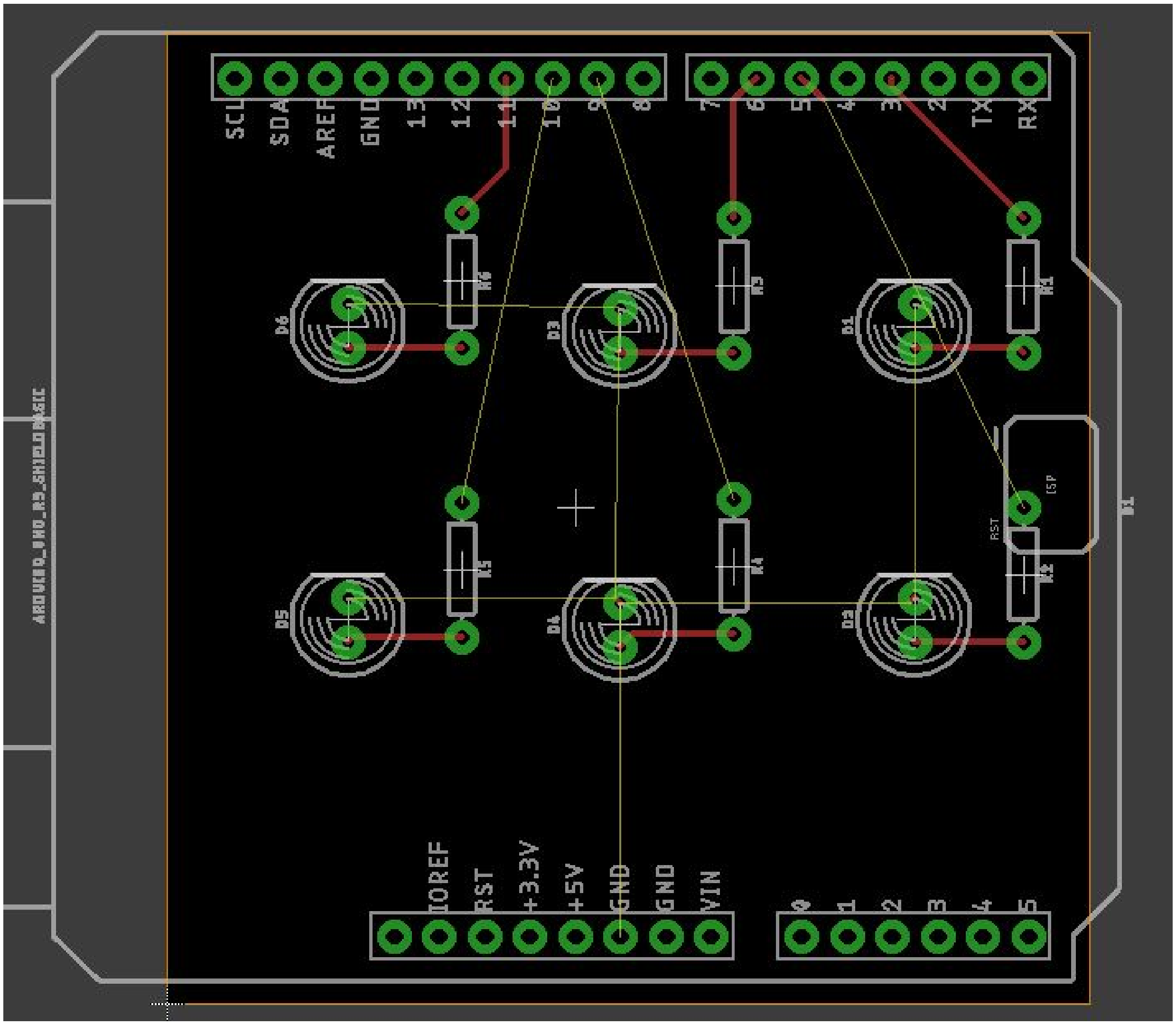
24. Connect R1, R3, and R6 to their arduino pins with the route tool (see example below).

a. Click inside a pad to start a trace and end on another pad.

25. Connect D6 to R6, D3 to R3, D1 to R1, D5 to R5, D4 to R4, and D2 to R2 with the route tool

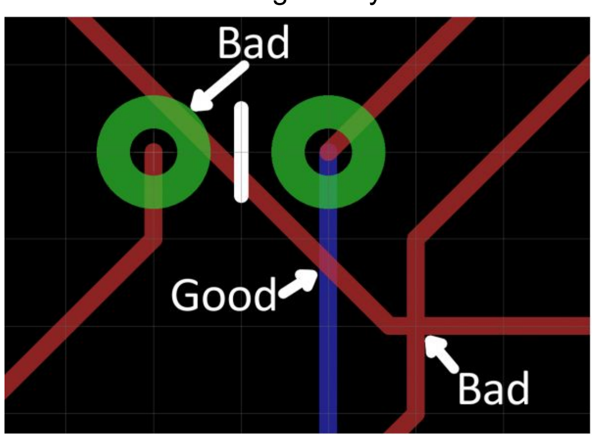
You can change trace directions at 45 degree angles. Be sure to keep some distance between the traces and the pins

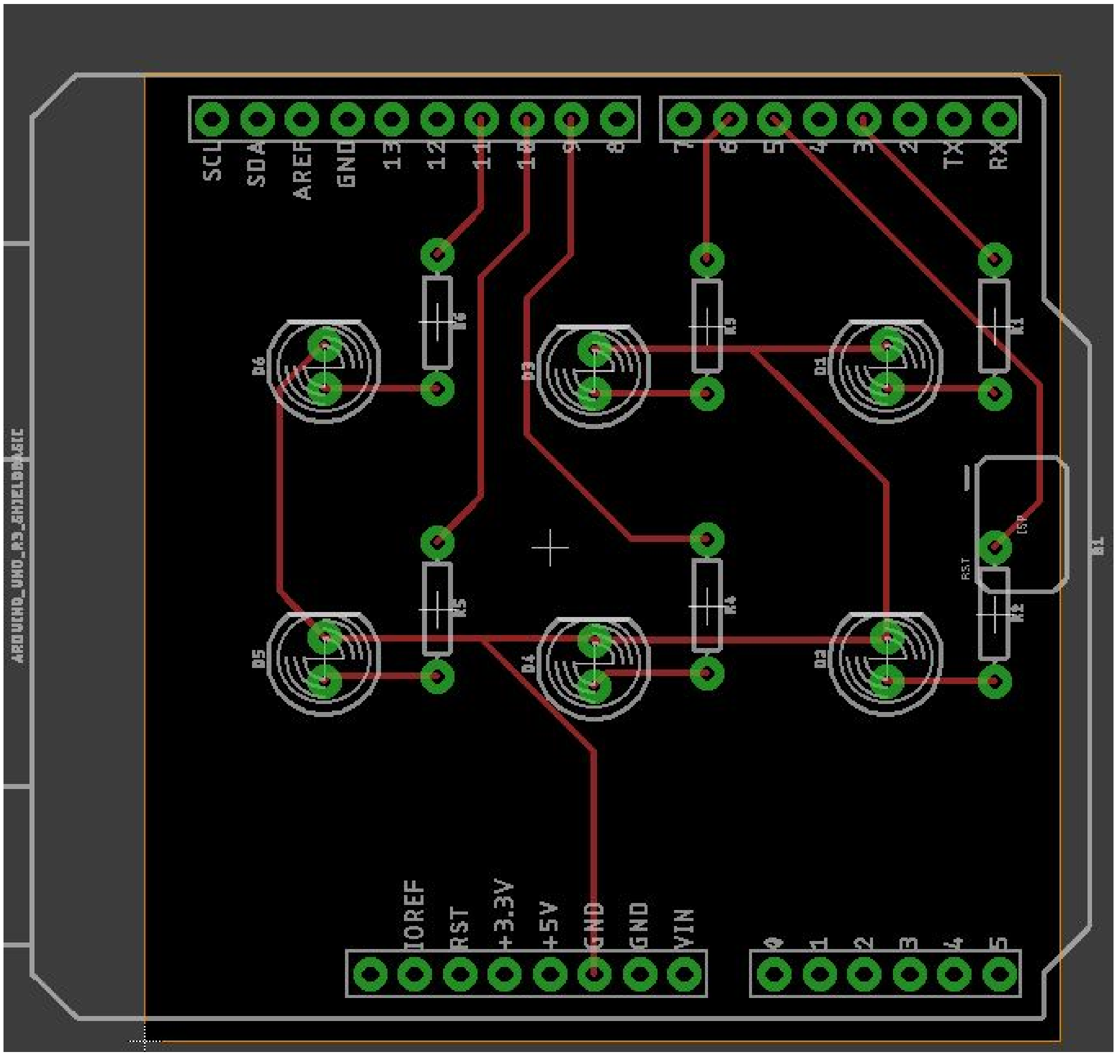
You can use the ripup tool to the right of the route tool to remove any traces you don't like. You can't use the delete tool to get rid of traces.



26. Strategically place the remaining traces. An example full board layout is shown below.
    a. Try to keep routing as direct as possible
    b. Don't cross your trace with another
    c. Avoid routing under components when possible

In the picture below, the different color traces (red and blue) represent traces on different layers of the board and therefore it's okay for them to cross. We're making a 1 layer board so none of our traces should cross.

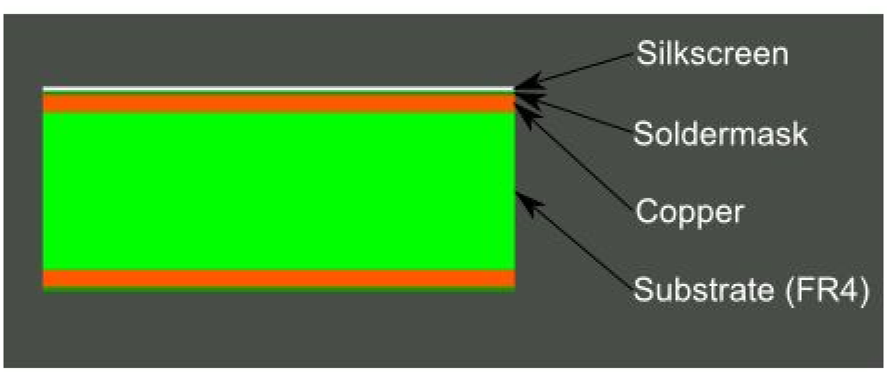




There should be no more visible airwires when you're done routing.

All the traces we just made will be on the top copper layer of the board. On top of that layer is the soldermask. This is what makes most PCBs green. The white writing on top of the soldermask is called the silkscreen.

Board Layers:

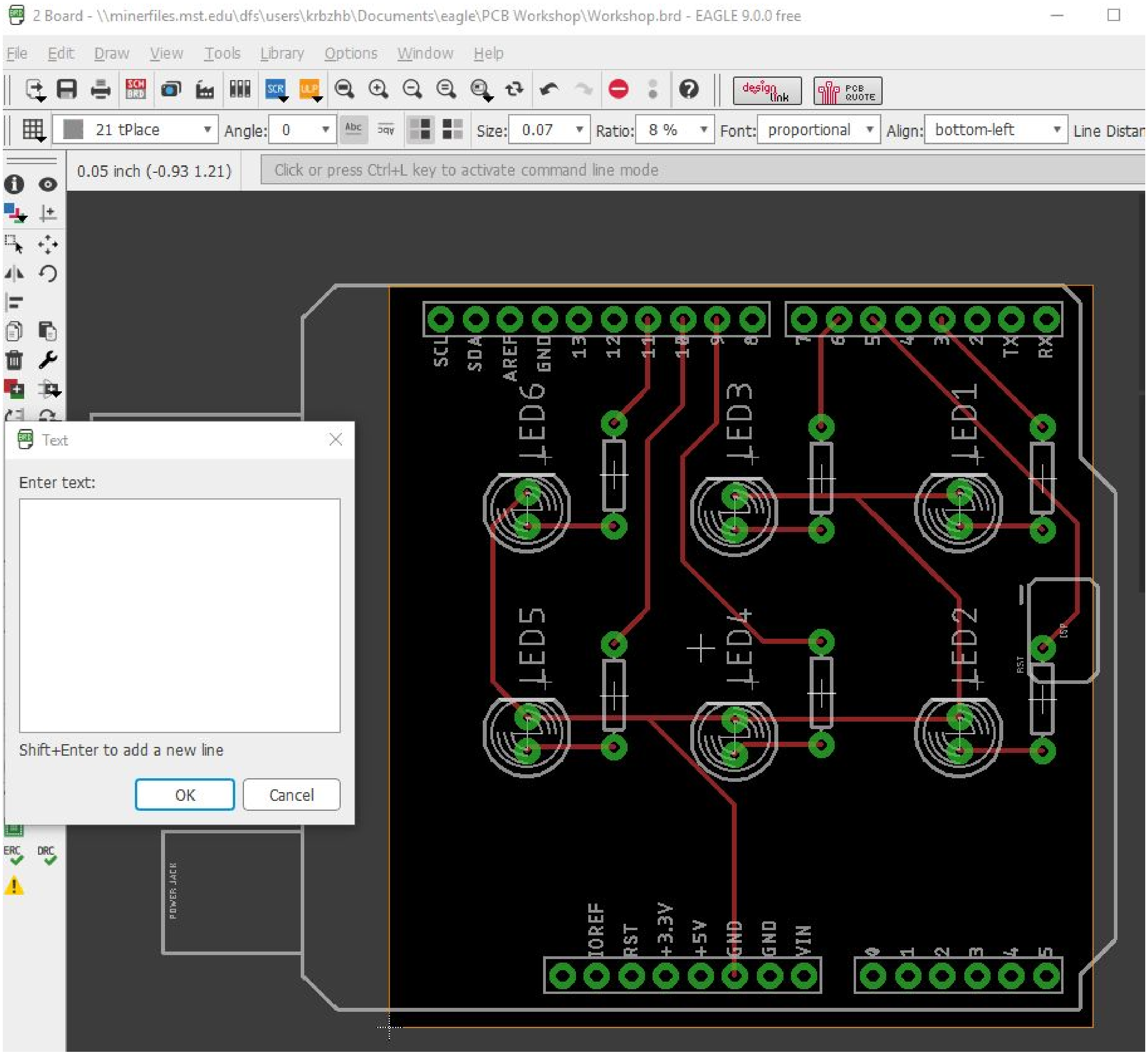


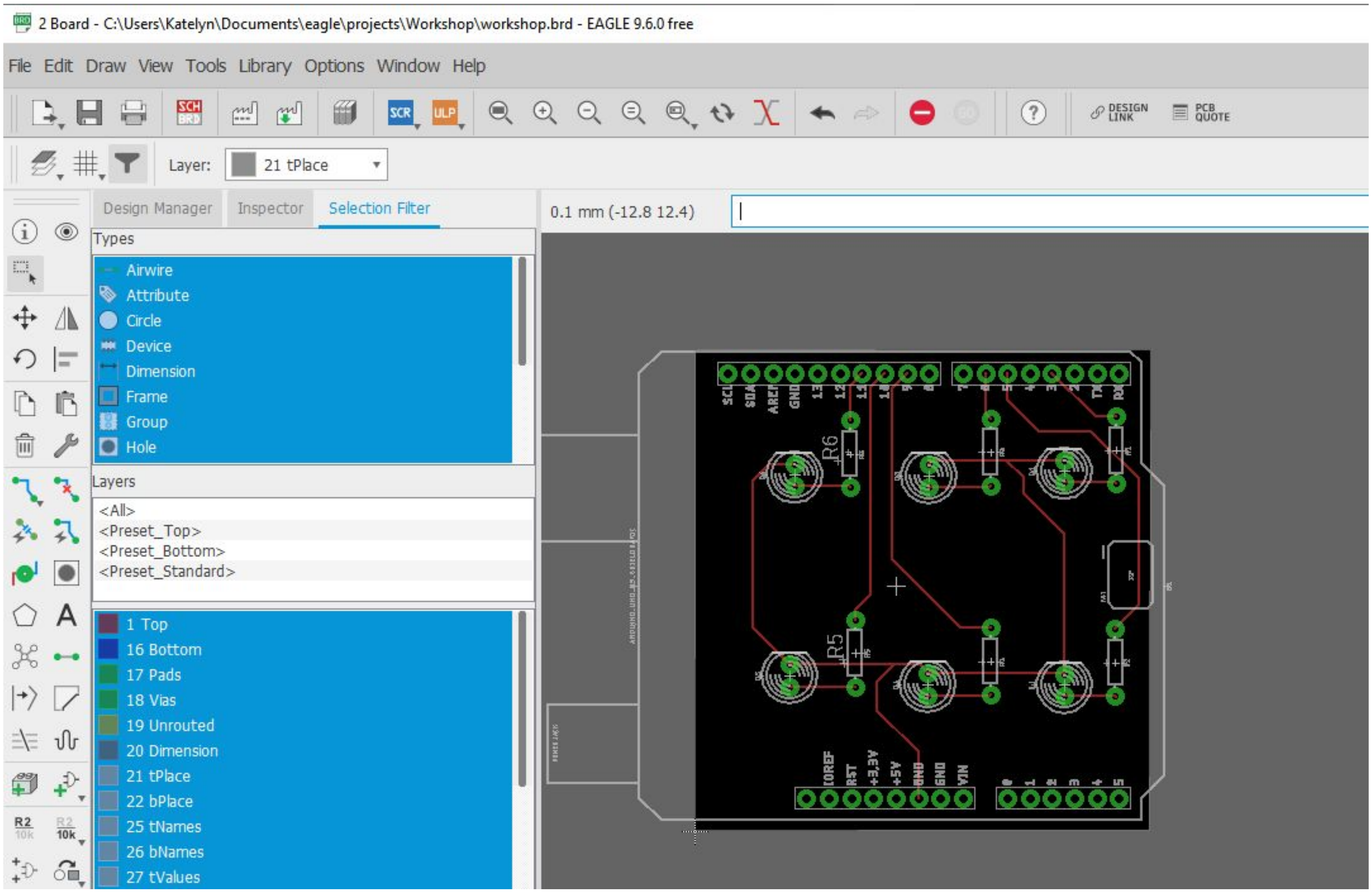
Color	Layer Name	Layer Number	Layer Purpose
	Top	1	Top layer of **copper**
	Bottom	16	Bottom layer of **copper**
	Pads	17	Through-hole pads. Any part of the green circle is exposed **copper** on *both* top and bottom sides of the board.
	Vias	18	Vias. Smaller copper-filled drill holes used to route a signal from top to bottom side. These are usually covered over by soldermask. Also indicates **copper** on both layers.
	Unrouted	19	Airwires. Rubber-band-like lines that show which pads need to be connected.
	Dimension	20	Outline of the board.
	tPlace	21	**Silkscreen** printed on the top side of the board.
	bPlace	22	**Silkscreen** printed on the bottom side of the board.
	tOrigins	23	Top origins, which you click to move and manipulate an individual part.
	bOrigins	24	Origins for parts on the bottom side of the board.
/ / Hatch	tStop	29	Top stopmask. These define where **soldermask** should *not* be applied.
\ \ Hatch	bStop	30	Absent **soldermask** on the bottom side of the board.
	Holes	45	Non-conducting (not a via or pad) holes. These are usually drill holes for stand-offs or for special part requirements.
	tDocu	51	Top documentation layer. Just for reference. This might show the outline of a part, or other useful information.

27. Add text on the silkscreen to label your components. Click on the text button. Type in your label. Click okay and then change the layer to 21 tPlace by finding it in the drop down menu at the top of the window before placing your label next to the correct component on the board

Typically we label our components on the silkscreen so we know what components to put where during assembly.

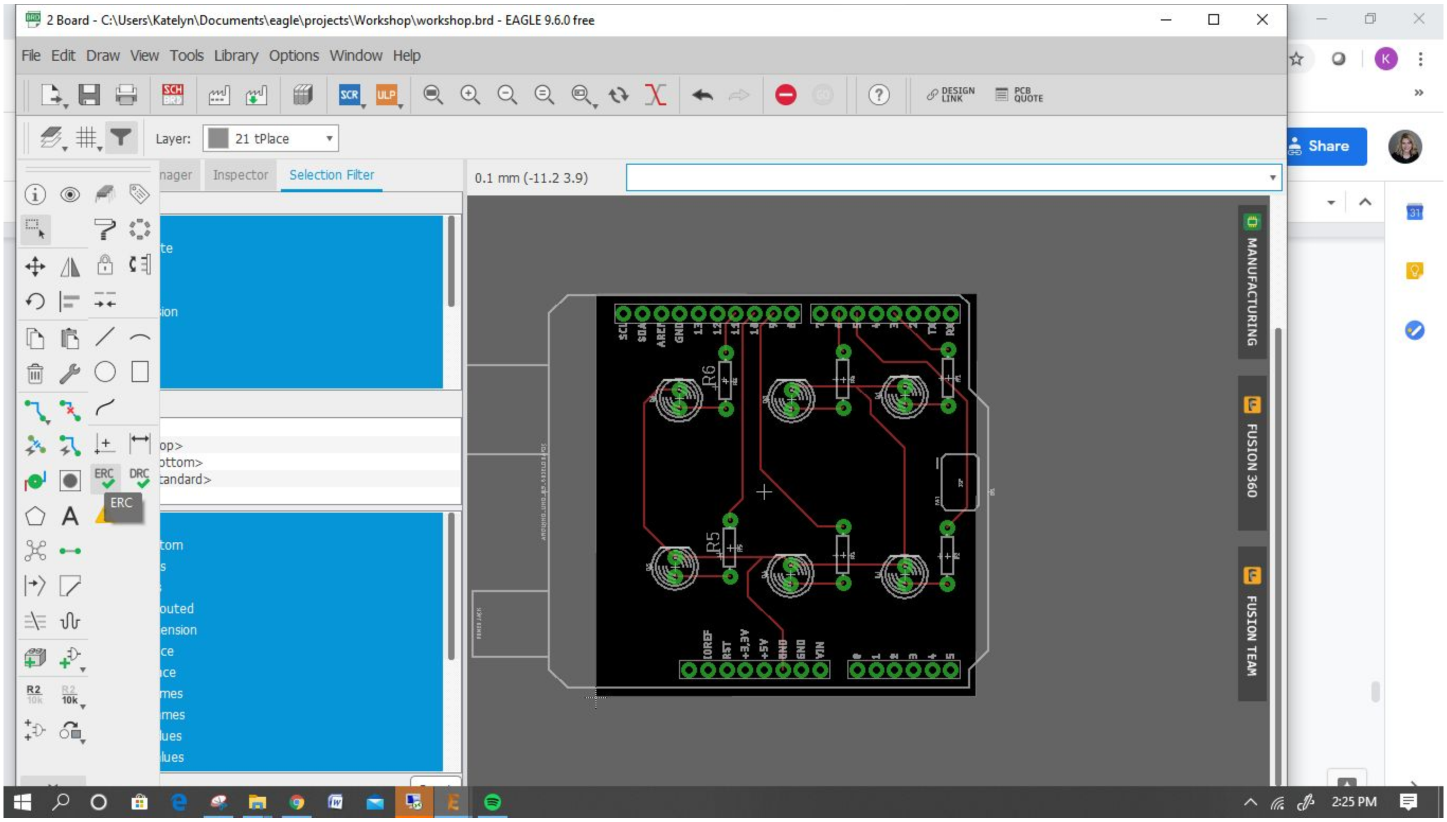
28. Save your board.


2 Board - \\minerfiles.mst.edu\dfs\users\krbzhb\Documents\eagle\PCB Workshop\Workshop.brd - EAGLE 9.0.0 free
File Edit Draw View Tools Library Options Window Help
21 tPlace
Angle: 0
Size: 0.07
Ratio: 8 %
Font: proportional
Align: bottom-left
0.05 inch (-0.93 1.21)
Click or press Ctrl+L key to activate command line mode
Text
Enter text:
Shift+Enter to add a new line
OK
Cancel
SCL SDA AREF GND 13 12 11 10 9 8 7 6 5 4 3 2 TX RX
LED6 LED3 LED1 LED5 LED4 LED2
IOREF RST +3.3V +5V GND GND VIN
0 1 2 3 4 5
POWER JACK

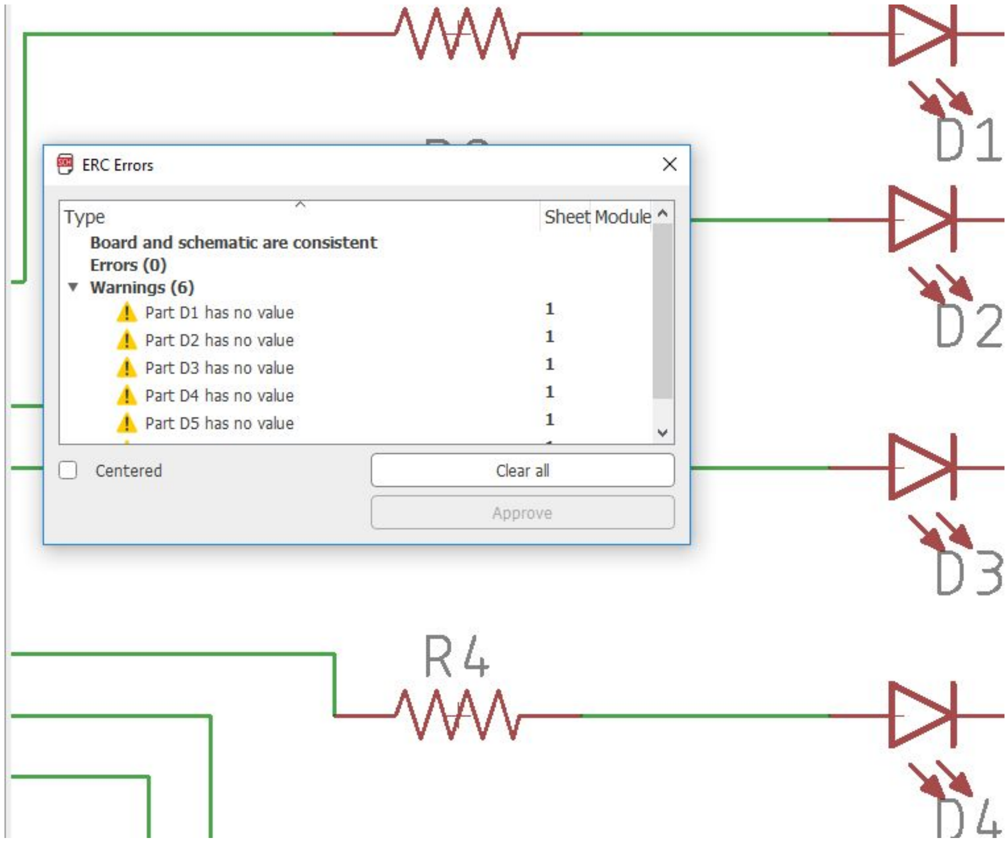

2 Board - C:\Users\Katelyn\Documents\eagle\projects\Workshop\workshop.brd - EAGLE 9.6.0 free
File Edit Draw View Tools Library Options Window Help
DESIGN LINK
PCB QUOTE
Layer: 21 tPlace
Design Manager
Inspector
Selection Filter
0.1 mm (-12.8 12.4)
Types
Airwire
Attribute
Circle
Device
Dimension
Frame
Group
Hole
Layers
<All>
<Preset_Top>
<Preset_Bottom>
<Preset_Standard>
1 Top
16 Bottom
17 Pads
18 Vias
19 Unrouted
20 Dimension
21 tPlace
22 bPlace
25 tNames
26 bNames
27 tValues
R6
R5
IOREF
RST
+3.3V
+5V
GND
GND
VIN

Finishing Up

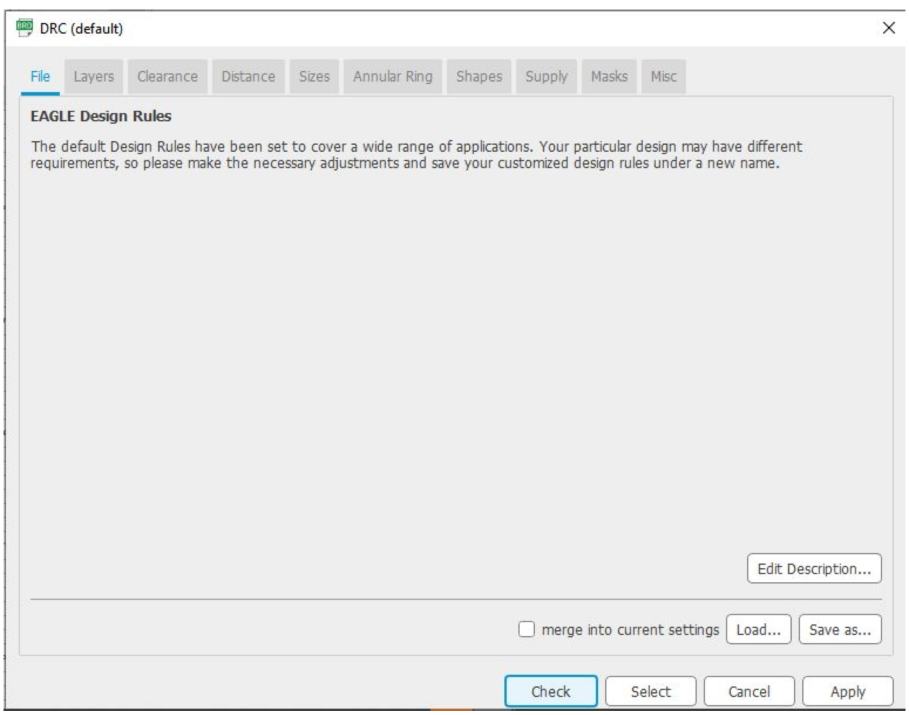
PCB design software has built in checks that you can run to make sure that a PCB manufacturer will be able to make your board. These checks look at things like if your traces are too close to pads or to each other.



29. Check your board design.Hit the ERC button at the bottom left of your toolbar. You might need to hit the down arrow at the bottom of the toolbar to show the other tools. You may see warnings about the LEDs not having values. You can clear these warnings.

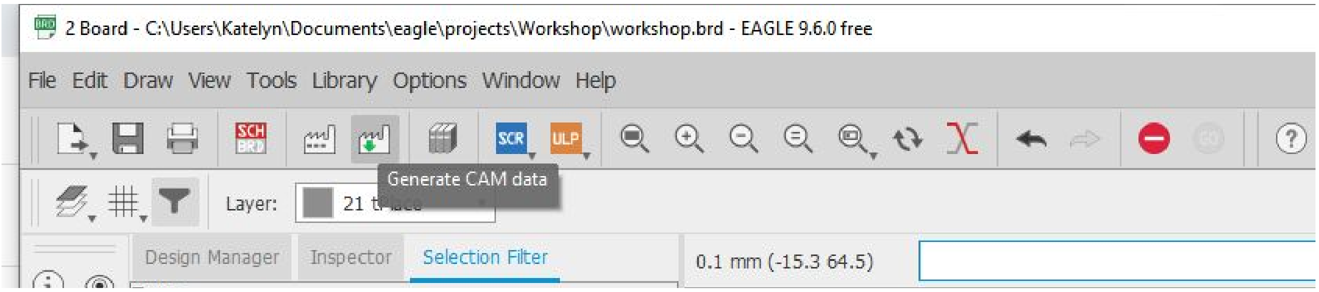


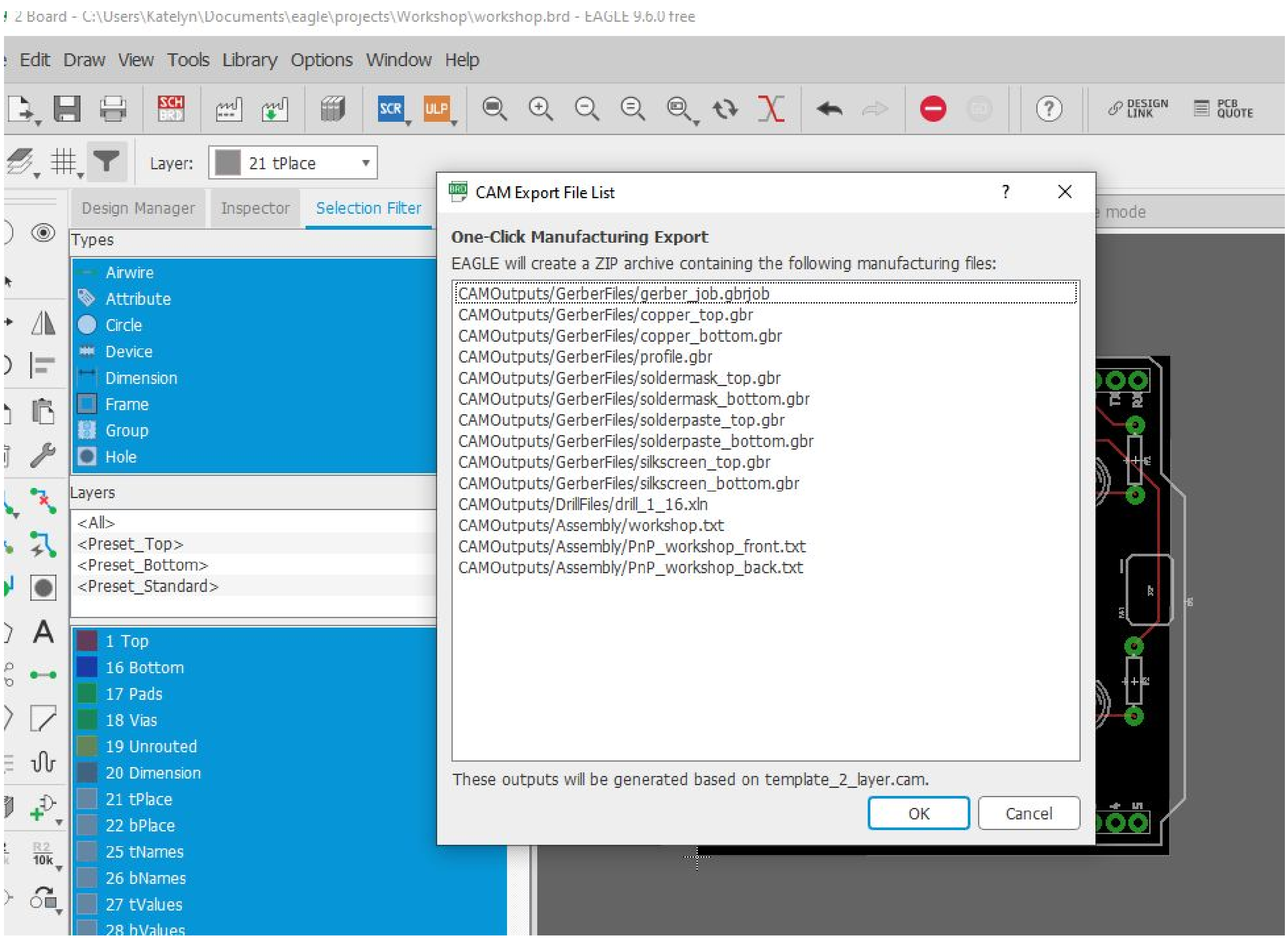
30. In the board layout view, in the DRC button next to the ERC button. This will check your layout for errors. Hit “Check” in the pop up window. This will run the default design rule check. PCB manufacturers also have their own DRCs that you can download and then run in your PCB design software.



1. Generate the gerber files. Click on the generate CAM data button. Then click okay.

Gerber files are what we send to PCB manufacturers to make boards. They provide the mechanical description of each board layer.





Additional Resources

Making your own Custom components:

There will be times when you can't find a pre-made library for a component that you need. In this case you need to use the datasheet to make it yourself. The following tutorial walks through the steps.

- https://www.autodesk.com/products/eagle/blog/library-basics-part-1-creating-first-package-autodesk-eagle/
- https://learn.adafruit.com/ktowns-ultimate-creating-parts-in-eagle-tutorial/adding-%3Ename-and-%3Evalue-to-your-symbol

Routing on 2 or more layers:

Sometimes, you can't route all of your traces without crossing other traces, which isn't allowed, or making really round about paths. In these cases, you can route on multiple layers (i.e., you can put traces on a top layer and a bottom layer).

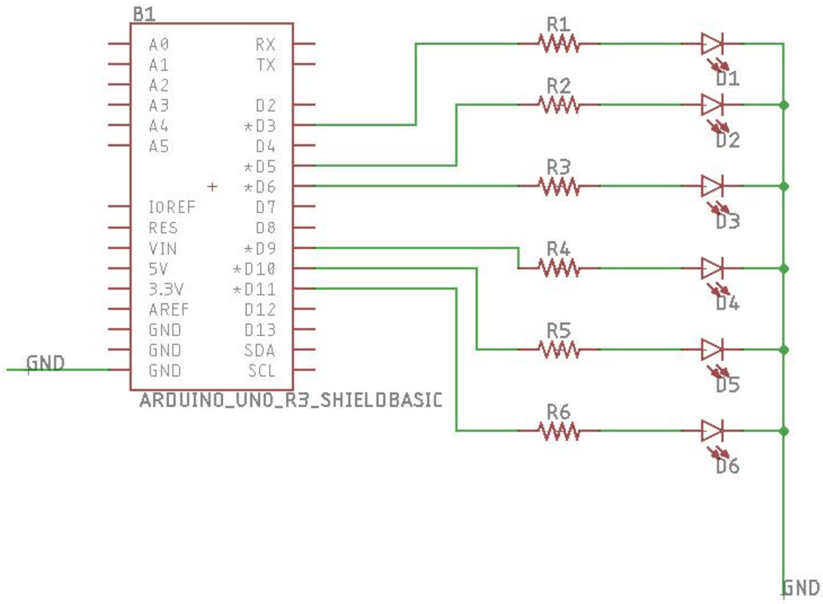
You can use vias to change layers.

Check out the following tutorials for more information.

- https://learn.sparkfun.com/tutorials/using-eagle-board-layout
- http://dangerousprototypes.com/blog/2012/07/18/eagle-polygons/

Enhancing your schematic:

Use the name tool to name nets and connect them without running a net wire between them. When you name two nets the same, Eagle will ask you if you want to connect them. This will clean up your schematic and make it easier during layout to understand what you're connecting.



You can also add values to your components in your schematic.

- https://www.autodesk.com/products/eagle/blog/schematic-basics-part-2-nets-and-values/
- https://learn.sparkfun.com/tutorials/using-eagle-schematic/tips-and-tricks

PCB Fabrication

In order to fabricate a PCB, you usually sent it to a fab house. Different fab houses have different requirements you must meet for them to fabricate your board. You can find these requirements on their website. Fab houses are have different costs and production times.

Here are some fab house options:

- https://easyeda.com/
- https://www.sunstone.com/
- https://www.4pcb.com/
- https://oshpark.com/

Additional Design Tutorials and Example Projects:

- https://learn.sparkfun.com/tutorials/designing-pcbs-advanced-smd
- https://learn.sparkfun.com/tutorials/arduino-shields
- https://circuitdigest.com/diy-pcb-projects