# Intro to PCB Design

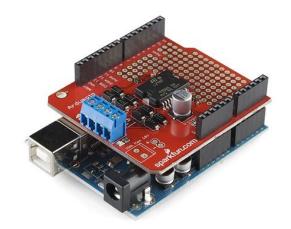
### Objectives

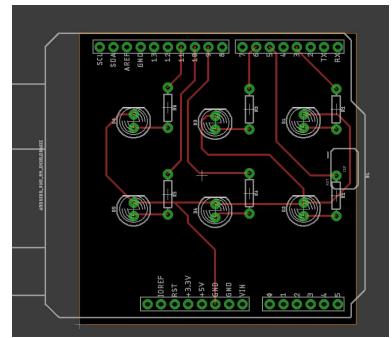
In this tutorial we'll design a printed circuit board (PCB) in Eagle. This PCB will be an Arduino shield - a board that sandwiches with an Arduino microcontroller to provide extra functionality. The PCB will have a collection of LEDs on it that can be controlled from the arduino.

- 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: <u>katelyn.brinker@ieee.org</u> or <u>brinker@iastate.edu</u>





# **Getting Started**

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

- Eagle
- Altium
- Cadence
- KiCAD
- CircuitMaker

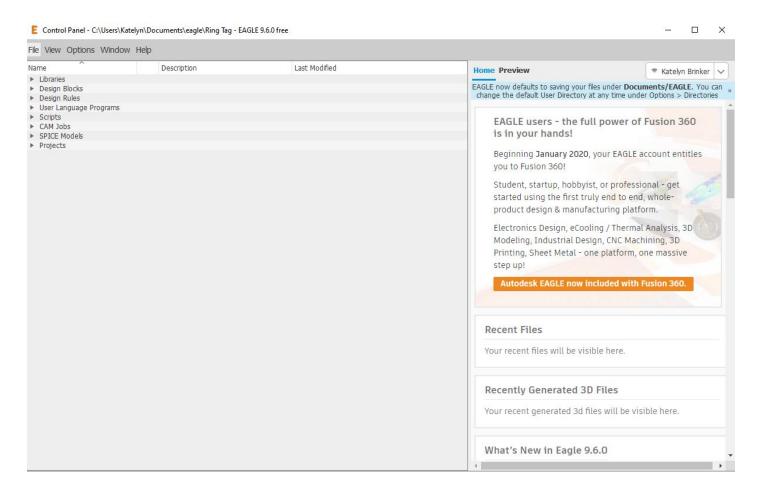
Eagle, KiCAD, and CircuitMaker have free versions. Altium and Cadence are both really powerful and customizable, but this makes them more difficult to learn on. The free version of Eagle is pretty user friendly and straightforward to learn on and provides the functionality we need for the workshop, so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

- 1. Download Eagle from the autodesk website: <u>https://www.autodesk.com/products/eagle/free-download</u>
- 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.

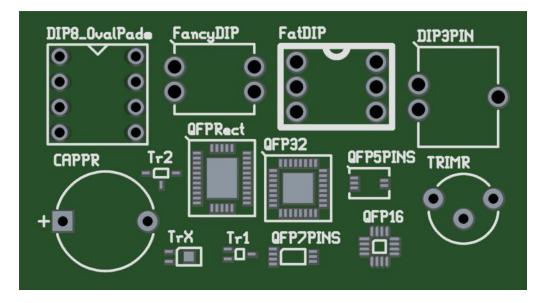
💽 Sign in			1 <u>00</u>	×
Options View				
	Sign in	۵		
	Email			
	hame@example.com			
	NEXT			
	NEW TO AUTODESK? CREATE ACCOUNT			

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.

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

-o- 2,270 commits		P 2 branches			ses 💄	52 contributors		
Branch: maste	r 🗸 New pull request				Find file	Clone or download		
igen edspark	Adss 0402 footprint for 1.0	nF Capacitor			Latest comr	nit 8d87d9d 4 days ago		
📄 images		Adding images for	wiki.	3 ує				
igitignore added		added in standard	gitignore stuff	4 years ag				
LilyPad-Wearables.lbr Updating library descriptions, mo		scriptions, moving Vout	Vout to PowerSymbol library 3					
SparkFun-Aesthetics.Ibr		Control:bit logo to	Controller:bit	3 months age				
SparkFun-Batteries.lbr		Add silk to SMD su	per cap.		7 months ag			
SparkFu	n-Boards.lbr	Add Artemis RedBo	ard compatible shields			2 months ago		
	ဖို 2 branches	🗇 <b>0</b> pack	ages	🛇 9 releases	1 52 contr	ibutors		
pull request					Find file Clone of	or download 🛨		
otprint for 1.0	)nF Capacitor			Clone with HTTPS (	9			
	Adding imag	ges for wiki.		Use Git or checkout with	URL.			
	added in sta	andard gitignore stuff		https://github.com/s	Eag. 🔂			
or	Updating lib	prary descriptions, movin	g Vout to PowerSym	Open in Desktop	Downlo	ad ZIP		
s.lbr	Control:bit	ogo to Controller:bit			3 r	nonths ago		
	و حد بالند الدام و				7 -			

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"

Control Panel - C:\Users\Katelyn\D	ocuments\eagle\Ring Tag - EA	GLE 9.6.0 free		- 🗆 X
le View Options Window Help				
	E Project	Last Modified	Home Preview	😤 Katelyn Brinker 🗸
	Schematic		EAGLE now defaults to saving your files change the default User Directory at a	
Open recent projects	Board		change the default User Directory at a	any time under Options > Directories
Close project	Library			
Exit Alt+X ( SPICE Models	Design Block			
Projects	CAM Job			
<ul> <li>projects</li> <li>examples</li> </ul>	ULP ULP			
	Script			
	Text			
Control Panel - C:	Users\Katelvn\D	ocuments\EAGLE\projects\	Workshop - EAGLE 9.6.0 free	
			F	
	Window Help			
le View Options				
~		Description	Lash Mar J.C. J	
ame		Description	Last Modified	
ame		Description	Last Modified	
ame Â		Description	Last Modified	
ame Libraries Design Blocks		Description	Last Modified	
ame Libraries Design Blocks Design Rules	grams	Description	Last Modified	
ame Libraries Design Blocks Design Rules User Language Pro	grams	Description	Last Modified	
ame Libraries Design Blocks Design Rules User Language Pro Scripts	ıgrams	Description	Last Modified	
ame Libraries Design Blocks Design Rules User Language Pro Scripts CAM Jobs	grams	Description	Last Modified	
ame Libraries Design Blocks Design Rules User Language Pro Scripts CAM Jobs	igrams	Description	Last Modified	
ame Libraries Design Blocks Design Rules User Language Pro Scripts CAM Jobs	igrams	Description	Last Modified	

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

Empty Project

Examples Projects

🔹 📱 Workshop

comples

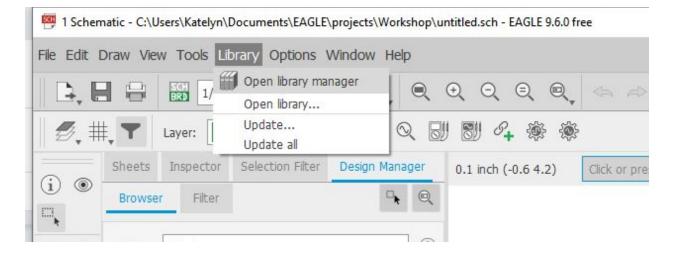
6. Right click on your project in the Control panel then select New - Schematic. A schematic will then be added to your project and will open in a new window.

ile View Options V					
lame	Descrip	otion	Last Modified	Home Preview	😤 Katelyn Brinker 🥆
<ul> <li>Libraries</li> </ul>					
<ul> <li>Design Blocks</li> </ul>				EAGLE now defaults to saving your file change the default User Directory at	es under Documents/EAGLE. You car
Design Rules					any time under Options > Directories
<ul> <li>User Language Prog</li> </ul>	grams			Empty Project	
<ul> <li>Scripts</li> </ul>				Use the context menu to create new	schematic or board files within this
CAM Jobs				project.	benefitiere en board meet mentri enb
SPICE Models					
Projects					
projects	User P				
🔹 📕 Worksho	Close Project	Project			
examples		s Projects			
	New 🕨				
	Rename	Board			
	Сору	Library			
	Delete	CAM Job			
	Edit Description	ULP			
	Use all	Script			
Use none					
		Text			
	Search in folder	Folder			
		Project			

🖣 1 Schematic - C:\Users\Katelyn\Documents\EAGLE\projects\Workshop\untitled.sch - EAGLE 9.6.0 free — 🗌 X	(
e Edit Draw View Tools Library Options Window Help	
一 # Layer: ■ 91 Nets ・ へ Q 副 副 ペ ● ●	
Sheets Inspector Selection Filter Design Manager Browser Filter Filter 0.1 inch (-0.0 -0.8)	•
View: Parts • ?	
Project Documents 1 of 1 shown (1 selected)	
Name Type       Name     Type       All Sheets	
Sheet 1/1 Sheet	
Parts 0 of 0 shown (0 selected)	
AB Type Name Value	
2 <u>R2</u> Dk 10k	
Tems 0 of 0 shown (0 selected)	
Y ∠ Q Search	
Type Name Parent	
	. 1

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.

Name     Version     Managed Folder       19inch     40xx       41xx     41xx       45xx     74ac-logic       74ac-logic     74xc-eu       74ttl-din     74xc-eu       74xc-eu     74xc-logic       74xx-us     75xx       751xx     advanced-test       agilent-technol     allegro       altera     altera       altera-cyclone-III     altera-cyclone-III       altera-cyclone-III     and-mach       amd     amd-mach		Pescription 19-Inch Slot Eurocards (C:/Program Files (x86)/EAGLE-6.6.0/lbr./ CMOS Logic Devices, 4000 Series (C:/Program Files (x86)/EAGL. 41xx Series Devices (C:/Program Files (x86)/EAGLE-6.6.0/lbr/41. CMOS Logic Devices, 4500 Series (C:/Program Files (x86)/EAGL. TTL Logic Devices, 4500 Series (C:/Program Files (x86)/EAGLE-6. TTL Devices with DIN Symbols (C:/Program Files (x86)/EAGLE-6. TTL Devices, 74xx Series with European Symbols (C:/Program Files (x86)/EAGLE-6. Single and Dual Gates Family, US symbols (C:/Program Files (x86. Single and Dual Gates Family, US symbols (C:/Program Files (x86. TTL Devices, 74xx Series with US Symbols (C:/Program Files (x86. TTL Devices, 74xx Series with US Symbols (C:/Program Files (x86. TTL Devices, 74xx Series with US Symbols (C:/Program Files (x86.) TTL Devices (C:/Program Files (x86)/EAGLE-6.6.0/lbr/7. Advanced Test Technologies - Phoenix, Arizona USA (C:/Program Agilent Technologies (C:/Program Files (x86)/EAGLE-6.6.0/lbr/7. Altera Programmable Logic Devices (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. Allegro MicroSystems, Inc (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. Altera Acyclone II FPGA (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. Altera Stratix IV (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. Altera Stratix IV (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. Advanced Micro Devices Flash Memories (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. AMD Packages (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. AMD Packages (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. AMD MACH4/MACH5 Family (Vantis) (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1. AMI Semiconductor (C:/Program Files (x86)/EAGLE-6.6.0/lbr/1.		
Delete Bundled Libraries 9-Inch Slot Eurocards			Remov	
00 x 160 mm with DIN 41612 connec ME-BUS-Cards - http://www.interface		/ME.html	Update	
reated by librarian@cadsoft.de			Edit	

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.

Organize Vew folder Quick access Desktop Documents Documents Documents Downloads Atium Arduino HFS Processing Coc	SparkFun-Eagle-Libraries-master >	<ul> <li>C</li> <li>Date modifie</li> <li>3/22/2020 9:3</li> </ul>	d 30 AM 30 AM 30 AM 30 AM 30 AM 30 AM 30 AM	Fun-Eagle-Librari Fun-Eagle-Librari Type EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library	م 2	^
Quick access     Desktop     Deptop     Dopbox     Sp     Documents     Sp     Documents     Sp     Downloads     Sp     4-14-19     Attium     Arduino     HFS Processing Coc     Sp	arkFun-Electromechanical arkFun-Fuses arkFun-Hardware arkFun-IC-Amplifiers arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Logic	3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3	80 AM 80 AM 80 AM 80 AM 80 AM 80 AM	Type EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library	3	^
<i>→</i> Quick access             Desktop <i>→ →</i>	arkFun-Electromechanical arkFun-Fuses arkFun-Hardware arkFun-IC-Amplifiers arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Logic	3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3	80 AM 80 AM 80 AM 80 AM 80 AM 80 AM	EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library		^
Desktop     ★     Sp       ➡ Dropbox     ★     Sp       ➡ Documents     ★     Sp       ➡ Pictures     ★     Sp       ➡ Downloads     ★     Sp       ➡ Attium     Sp       ➡ Arduino     Sp       ➡ HFS Processing Coc     Sp	arkFun-Electromechanical arkFun-Fuses arkFun-Hardware arkFun-IC-Amplifiers arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Logic	3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3	80 AM 80 AM 80 AM 80 AM 80 AM 80 AM	EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library		
Desktop ★ Sp ↓ Dropbox ★ Sp → Documents ★ Sp → Downloads ★ Sp ↓ 4-14-19 Sp ↓ Altium Sp ↓ Arduino Sp ↓ HFS Processing Coc	arkFun-Electromechanical arkFun-Fuses arkFun-Hardware arkFun-IC-Amplifiers arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Logic	3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3	30 AM 30 AM 30 AM 30 AM 30 AM	EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library		
Bocuments       ★       Sp         Pictures       ★       Sp         Downloads       ★       Sp         4-14-19       Sp         Altium       Sp         Arduino       Sp         HFS Processing Coc       Sp	arkFun-GPS arkFun-Hardware arkFun-IC-Amplifiers arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Logic	3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3	80 AM 80 AM 80 AM 80 AM	EAGLE library EAGLE library EAGLE library EAGLE library EAGLE library		
▶ Pictures     ★     ▶ Sp       ▶ Downloads     ★     ▶ Sp       ↓ 1.4-19     ↓ Sp       ↓ Altium     ↓ Sp       ↓ Arduino     ↓ Sp       ↓ HFS Processing Coc     ↓ Sp	arkFun-Hardware arkFun-IC-Amplifiers arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Memory	3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3	80 AM 80 AM 80 AM	EAGLE library EAGLE library EAGLE library EAGLE library		
↓ Downloads     ★     ↓     Sp     ↓-14-19     ↓     Altium     Sp     Arduino     HFS Processing Coc     Sp	arkFun-IC-Amplifiers arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Memory	3/22/2020 9:3 3/22/2020 9:3 3/22/2020 9:3	80 AM 80 AM	EAGLE library EAGLE library		
4-14-19 Sp Altium Sp Arduino Sp HFS Processing Coc	arkFun-IC-Comms arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Memory	3/22/2020 9:3 3/22/2020 9:3	0 AM	EAGLE library		
Altium Sp Arduino Sp HFS Processing Coc Sp	arkFun-IC-Conversion arkFun-IC-Logic arkFun-IC-Memory	3/22/2020 9:3		the second s		
Artium Sp Arduino Sp HFS Processing Coc	arkFun-IC-Logic arkFun-IC-Memory		80 AM	EACLE Element		
Arduino Sp HFS Processing Coc Sp	arkFun-IC-Memory	3/22/2020 9:3		EAGLE library		
HFS Processing Coc			30 AM	EAGLE library		
S Sp		3/22/2020 9:3	30 AM	EAGLE library		
	arkFun-IC-Microcontroller	3/22/2020 9:3	BO AM	EAGLE library		į.
😻 dropbox-Namespace 📃 Sp	arkFun-IC-Power	3/22/2020 9:3	BO AM	EAGLE library		
OneDrive	arkFun-IC-Special-Function	3/22/2020 9:3	BO AM	EAGLE library		
E Sp	arkFun-Jumpers	3/22/2020 9:3	BO AM	EAGLE library		
inis re	arkFun-LED	3/22/2020 9:3		EAGLE library		+
A Maturali	arkFun-PowerSymbols	3/22/2020 9:3		EAGLE library		
= sp	arkFun-Resistors	3/22/2020 9:3		EAGLE library		
	arkFun-Retired	3/22/2020 9:3		EAGLE library		
<u> </u>	arkFun-RF	3/22/2020 9:3		EAGLE library		ho
	arkFun-Sensors	3/22/2020 9:3		EAGLE library		
	arkFun-Switches	3/22/2020 9:3		EAGLE library		ia
Us K	er-Submitted	3/22/2020 9:3	IO AM	EAGLE library	>	Y dit
File name: "Us	er-Submitted" "LilyPad-Wearables" "Sp	oarkFun-Aes ~	Libraries (*.I	br)	~	
			Open	<b> </b> ▼ Cance	ł	

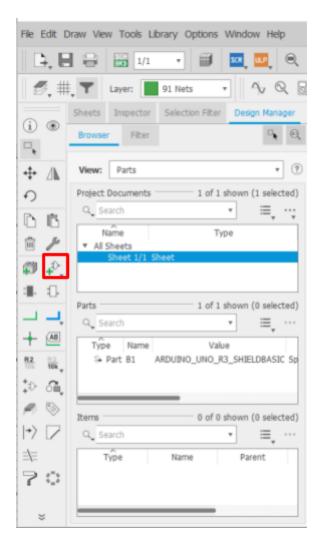
10. In the library manager under the "In Use" tab, scroll down and make sure the SparkFun libraries appear in the list.

Name	Version	Managed Folder	Description
sipex	V CI SION	Hanagea Folder	Sipex CorporationPower Management, Interface, Optical Storag
smd-ipc			IPC Standard SMD Packages (C:/Program Files (x86)/EAGLE-6.6
smd-special			Special SMD Packages (C:/Program Files (x86)/EAGLE-6.6.0/lbr/
SolarCar conn.			(C:/Users/Katelyn/Desktop/EAGLE-Libraries-Hardware-master/So
SolarCar_conn.			(C:/Users/Katelyn/Desktop/EAGLE-Libraries-Hardware-haster/So (C:/Users/Katelyn/Documents/eagle/EAGLE-Libraries-Hardware
SolarCar powe			(C:/Users/Katelyn/Documents/eagle/EAGLE-Libraries-Hardware-master/So
SolarCar powe			(C:/Users/Katelyn/Documents/eagle/EAGLE-Libraries-Hardware
solomon-syste			Solomon Systech Limited (C:/Program Files (x86)/EAGLE-6.6.0/l
solpad	cir:		Solder Pads/Test Points (C:/Program Files (x86)/EAGLE-6.6.0/b
SparkFun-Aest			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Batt			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Boar			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Buzz			(en obeig insteining oceaniente) openn en Euge Ebiente motern
SparkFun-Capa			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Clock			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Coils			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Conr	1		(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Discr			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Displ			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Elect	t		(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-Fuse			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/
SparkFun-GPS			(C:/Users/Katelyn/Documents/SparkFun-Eagle-Libraries-master/

# 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.

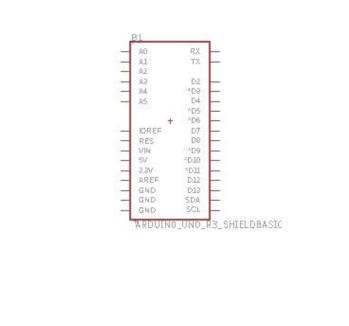
We'll use the tools in the toolbar on the left to build the schematic. The button to add components is boxed in red in the image below. Hovering over this tool brings up the description "Add Part"



- 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.

ADD	×
Name Managed	i Folder De
SolarCar_power_and_LDOs	
<ul> <li>SolarCar_power_and_LDOs</li> <li>SolarCar_power_and_LDOs</li> </ul>	
<ul> <li>solarca_power_and_coos</li> <li>solomon-systech</li> </ul>	
<ul> <li>solonion-system</li> <li>solpad</li> </ul>	
<ul> <li>SparkFun-Aesthetics</li> </ul>	Sp Sp H
<ul> <li>SparkFun-Batteries</li> </ul>	S0 A 100 A 1
<ul> <li>SparkFun-Boards</li> </ul>	Sp - w voia -
ARDUINO_MEGA_R3FULL	Ari - 4825 012
ARDUINO_PRO_MINI	
ARDUINO_UNO_R3	- GND SG -
ARDUINO_UNO_R3_SHIELD	
ARDUINO_UNO_R3_SHIELDBASIC	
ARDUNO_UNO_R3_SHIELDBASIC	UN ARDUINO_UNO_R3_SHIELD
ARDUNO_UNO_R3_SHIELDNOLABELS	
ARDUNO_UNO_R3_SHIELDUNO_R3_SHIELD_N	
ARDUINO_UNO_R3_SHIELD_ICSP	Arduino R3 Shield Footprint
<ul> <li>ARTEMIS_REDBOARD</li> </ul>	Shield form compatible with the
ATLAS BNC	Arduino Uno R3.
ATLAS CONDUCTIVITY	Aldalio olio ito.
► BIG_RED_BOX-PCB	Bic SparkFun Products:
BREADBOARDMINI	
CAN485FULL_SOCKET	Sp • <u>MG2639 Cellular Shield</u> • MIDI Shield
CAN485_OUTER_PINS_ONLY	R3 Stackable Headers Kit
CAN485_PRO_MINIPRO_MINI_COMPATIBLE	
	Ele Attribute Value
ELECTRIC_IMP_IMP002	
ESP32_THING MRED_NVR_LBC1760	Sp
MBED_NXP_LPC1768	mt
MICRO-SDCARD	mi
MICROSD-BREAKOUT	
🖓 Pads 🔍 Smds 🖃 Description 🗌 Hide Unpopular Parts	Preview
Search 🛛	×
Attributes 🛽	<b>*</b>
	OK Open Library Manager Cancel

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.



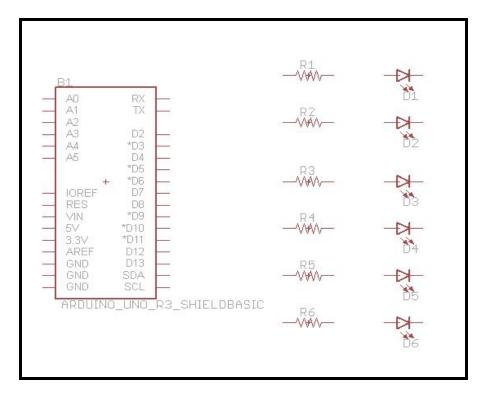
To pan the schematic canvas, hold down your mouse's scroll wheel and drag the canvas. To zoom in, roll the scroll wheel away from you. To zoom out, roll the scroll wheel towards you.

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.

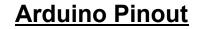
14. Place the components in the BOM below in the schematic. The Library column tells you where to find the part in the Add window. Remember that you can right click before placing a component to rotate it. After placing components, you can use the move tool in the left hand toolbar to arrange the symbols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

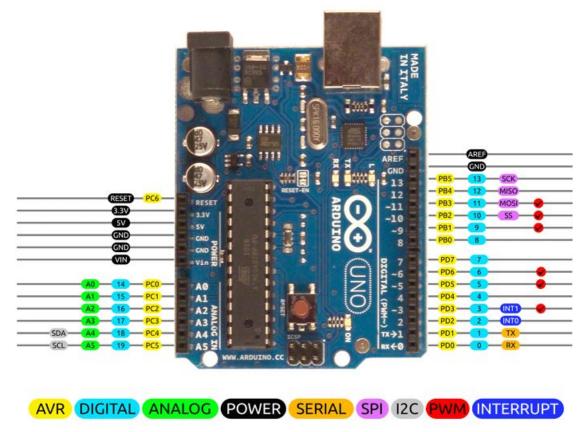
	<b>x y</b>	
Part Name	Library/Folder	Quantity
ARDUINO_UNO_R3_SHEILDBA SIC	SparkFun-Boards -> ARDUINO_UNO_R3_SHIELD	1
RESISTORAXIAL-0.3	Sparkfun-Resistors ->RESISTOR	6
LED5MM	Sparkfun-LED ->LED	6

### Bill of Materials (BOM)



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.



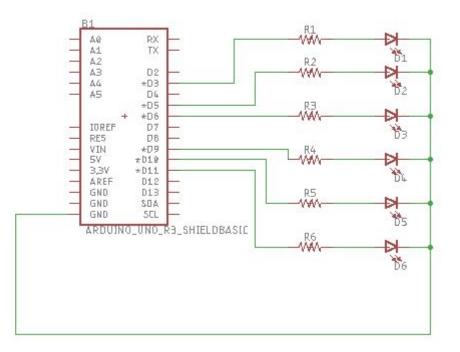


- 15. Use the Net tool (green L shape button) in the toolbar to connect the symbols according to the suggested connections and suggested schematic below. To use the Net tool, click on the pin of a symbol to start the net and then click on the pin of the component you want to connect to.
- 16. Save your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

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

### **Suggested Board Connections**

### **Suggested Schematic**

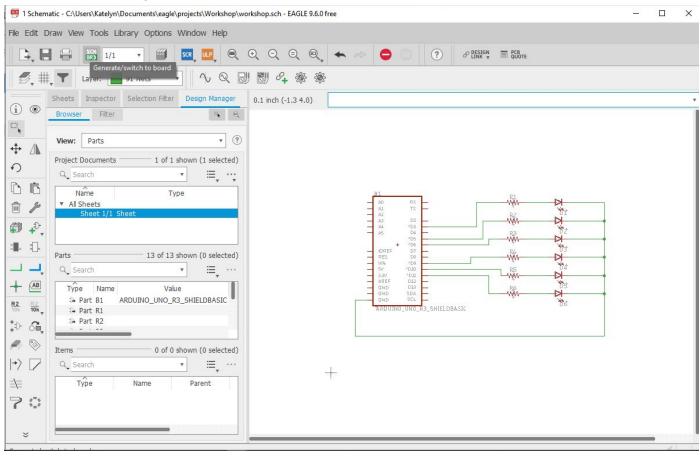


NOTE: There are many possible ways to layout the schematic (i.e., different pin connections and organizations of symbols). This schematic layout will make laying out the board easy and is therefore suggested.

### **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.

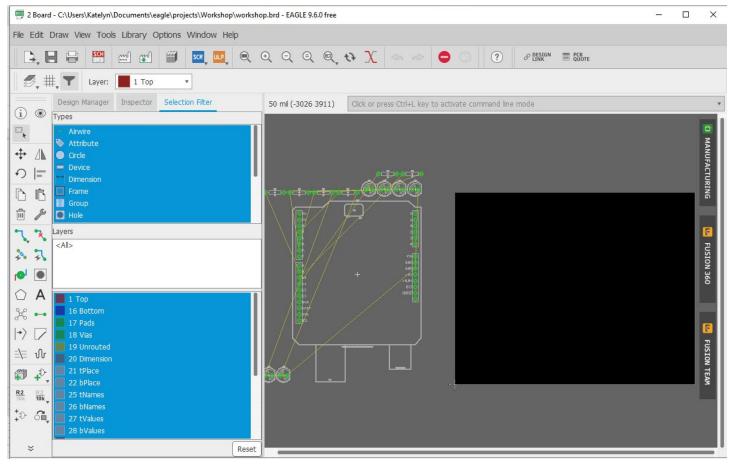


1	🥶 1 Schem	natic - C:\	Users\Katel	yn\Docum	ients\eag	e\projects	Works	shop∖v	/orksho	op.sch	- EAGL	E 9.6.0 f	ree
e	File Edit D	Draw V	ew Tools	Library	Options	Windo	w He	lp					
	D. E		SCH BRD 1	/1 *		SCR	ULP		Ð	Q	۵,	0,	+
	Ø, 1	🕶 Warn	ing	_				~ ~			、	×	
	(i) (i)		The board				s\eagle	e\proje	cts\W	orksho	р		Click
			Create fro			-*							
									Yes		N	•	
	۰ <sup>۲</sup>	Q,	Search			Ŧ	≡_		-				
	D B	-	Name		2	Гуре							
	â 🎤	▼ A	Sheets Sheet 1/	1 Sheet									
	en .D-												

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.

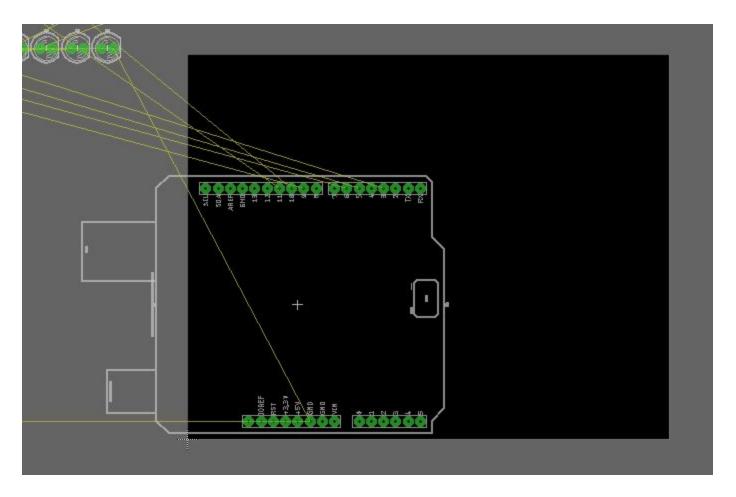
The black area in the board window is the board area. We can adjust the size and shape of the board area based on what we want to fabricate.



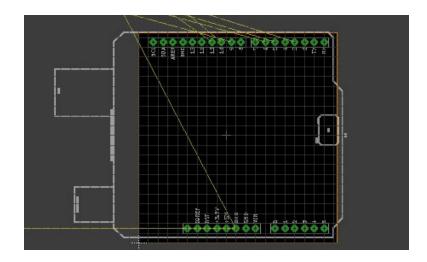
18. Adjust your grid - this is a personal preference. The grid will determine how you can move components. You can choose whether or not to display it.

🖳 Grid			×
Display	⊖ off	Style  O Dots  O Lines	
Size:	0.1	inch 🔻 Finest	
Multiple:	1		
Alt:	1	mm 🔻 Finest	
Defau	lt	OK Cancel	

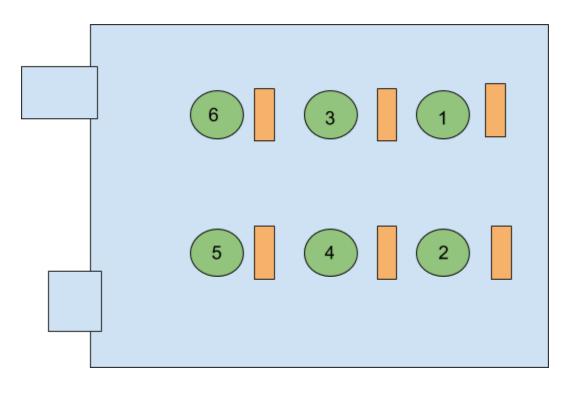
19. Move the Arduino shield into the board area using the move tool. Once you select the shield with the move tool (click on the plus in the middle of the shield outline) you can right click to rotate the shield before placing it. We're going to let the USB and ethernet ports hang off the edge. Since 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.

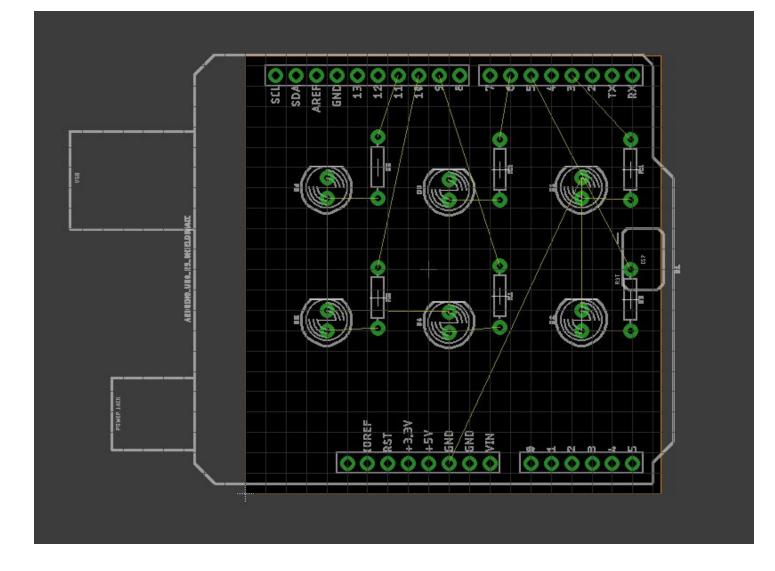


20. Adjust the board size using the move tool so it fits the profile of the arduino. With the move tool selected, click on an edge of the board and drag it in towards the shield. You can also right click on an edge of the board and select properties. In the properties window you can adjust the position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

File Edit Draw View

(i

Ш,

111

۲

L

Design Man

Types

8

Layers

<All>

Ratsnest

v

Ð

10k

=/=

**R2** 

17 Pads

18 Vias

19 Unro

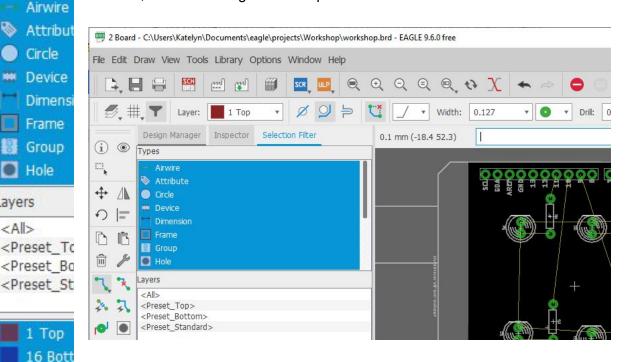
20 Dim 21 tPla

22 bPla

25 tNar

26 bNa 27 tVa

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.



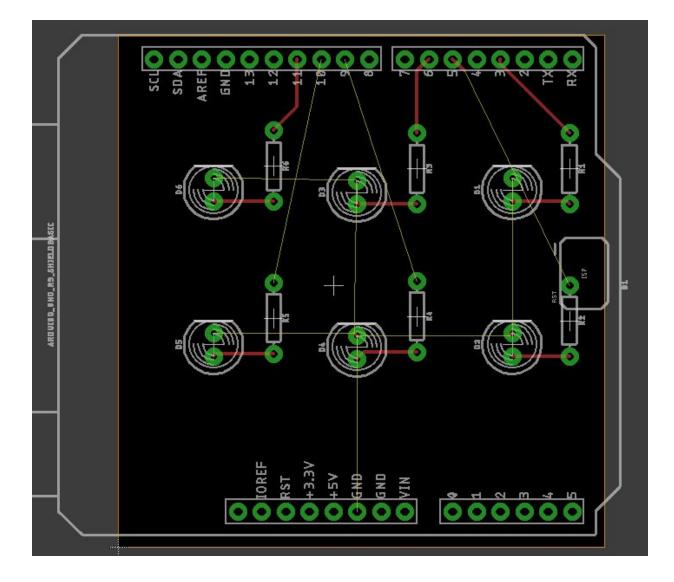
24. Connect R1, R3, and R6 to their arduino pins with the route tool (see example below).

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

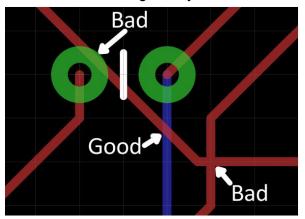
You can change trace directions at 45 degree angles. Be sure to keep some distance between the traces and the pins

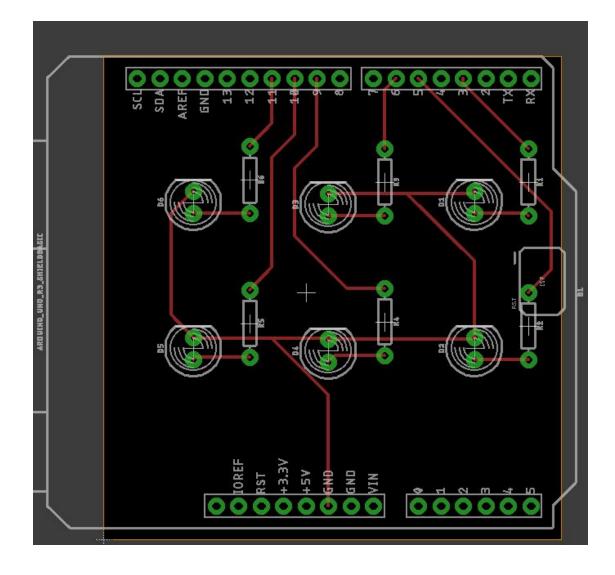
You can use the ripup tool to the right of the route tool to remove any traces you don't like. You can't use the delete tool to get rid of traces.



- 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.

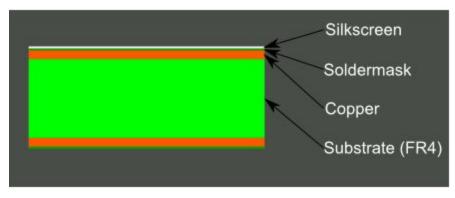




There should be no more visible airwires when you're done routing.

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:**



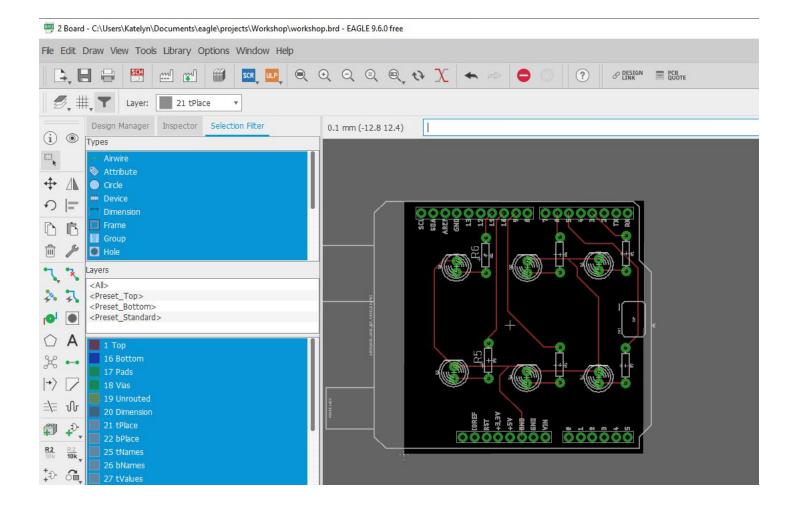
Color	Layer Name	Layer Number	Layer Purpose
	Тор	1	Top layer of copper
	Bottom 16		Bottom layer of copper
	Pads	17	Through-hole pads. Any part of the green circle is exposed <b>copper</b> on <i>both</i> top and bottom sides of the board.
	Vias	18	Vias. Smaller copper-filled drill holes used to route a signal from top to bottom side. These are usually covered over by soldermask. Also indicates <b>copper</b> on both layers.
	Unrouted	19	Airwires. Rubber-band-like lines that show which pads need to be connected.
	Dimension	20	Outline of the board.
1	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 <b>soldermask</b> should <i>not</i> be applied.
۱۱ 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.

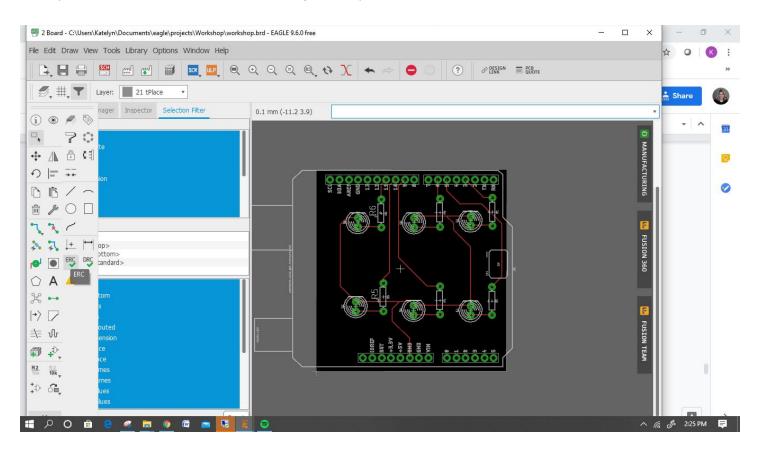
28. Save your board.

<u>File Edit Draw View Tools Library Options</u>	Window Help
Angle: 0 • Abc or	Size: 0.07 🔻 Ratio: 8 % 🔻 Font: proportional 🔻 Align: bottom-left 💌 Line Distant
0.05 inch (-0.93 1.21) Click or press Ctrl	+L key to activate command line mode
<b>₩</b> 12	
=	
	S IS A A I
🖗 Text 🛛 🗙	
Enter text:	
Shift+Enter to add a new line	
OK Cancel	
ERC DRC	
D A D	COREF SST SST SST SST SST SST SST SST SST SS

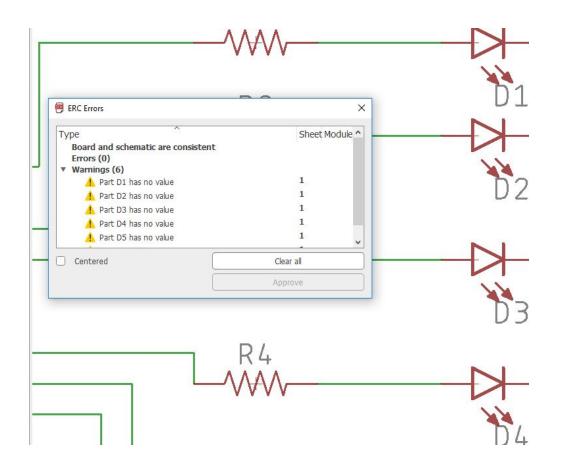


# Finishing Up

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.

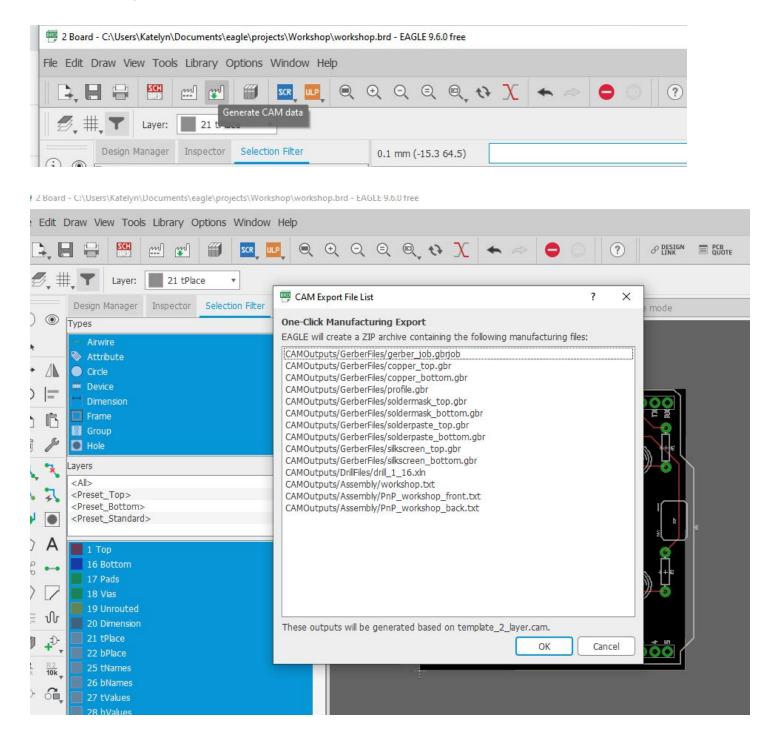


30. In the board layout view, in the DRC button next to the ERC button. This will check your layout for errors. Hit "Check" in the pop up window. This will run the default design rule check. PCB manufacturers also have their own DRCs that you can download and then run in your PCB design software.

		Clearance	Distance	Sizes	Annulas Dina						
	F Design				Annular Ring	Shapes	Supply	Masks	Misc		
he d	e ocoigii	Rules									
oquin	efault De	sign Rules ha	we been set	to cove	r a wide range o ustments and sa	of applicatio	ins. Your p	oarticular (	design may	have different	
equir	ements, s	so please mai	ke the nece	ssary adj	ustments and sa	ive your cu	stomized (	aesign ruie	es under a r	iew name.	
										Edit De	scription.
								1010			2
							merge	e into cun	rent setting	s Load	Save as

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.

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

#### 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).

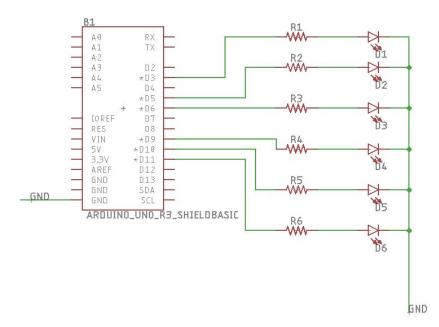
You can use vias to change layers.

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/
- <u>https://learn.sparkfun.com/tutorials/using-eagle-schematic/tips-and-tricks</u>

#### **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/
- <u>https://oshpark.com/</u>

#### Additional Design Tutorials and Example Projects:

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