Making a Circuit Board

Build your own custom circuit board in an evening

Charles McKinnis KU5N

August 2019

Outline

- Download and install Autodesk Eagle PCB design software,
- Create a schematic in Eagle,
- Create a printed circuit board (PCB) design,
- Etch the circuit board,
- Prepare the board for components, and
- Mount and solder the components.

Videos of the AutoDesk Eagle Design design steps are available on the YouTube.com KU5N Channel
https://www.youtube.com/playlist?list=PLIvKeVmi3PzRwqligRhgJHaMD6LFL_dT7

Downloading Autodesk Eagle

https://www.autodesk.com/products/eagle/overview

“FREE DOWNLOAD” at the top,

Available for Windows, Mac and Linux.

Free Version for hobbyists is limited:

- 2 schematic sheets,
- 2 signal layers,
- 80cm^2 (12.4in^2) board area for each project, and
- Personal, non-commercial use.

Toner transfer important notes

- This method involves transferring the toner a laser printer or copier uses to a copper clad board.
- This method will not work with an ink jet
  - It doesn't use toner and you can't heat transfer the dried liquid ink.
- This method will not work with a color laser printer.
  - The toner in color laser printers has a different granular size.
  - Learn from my personal experience and frustration, the print will look acceptable, but the transfer will be "fuzzy" and unusable.
- The library lets you print 10 pages a day for free, just ask to use your own "special paper" in the manual feed tray.
- Or, print with an ink jet, and photocopy the page to a magazine sheet.
  - Modern black & white copiers are laser printers.

Starting a New Project

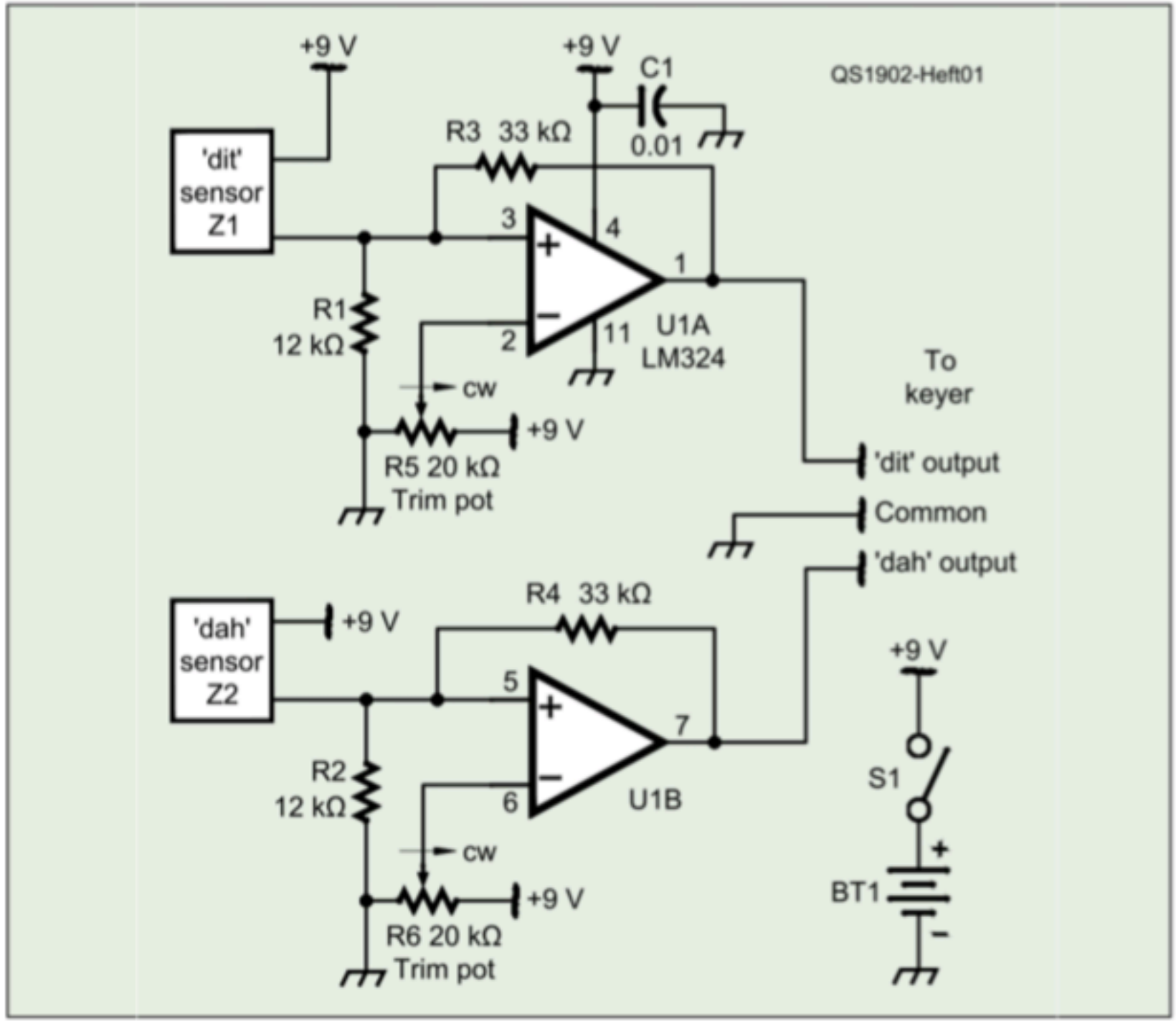
- Load Eagle, it opens to the Control Panel,
- Find Projects on the left, and expand it,
- Right click the project folder and create a new project,
- Right click the project you created, and choose Open Project,
- Right click the project, click New, click Schematic.

Video 01 https://youtu.be/ZMCfNcSNpzM

Download the component libraries

- In the Drop Down menus, click Library, Open Library Manager
- Select the Available tab,
- Select the first item, scroll to the bottom, hold shift and select the last item,
- Click the “Use” button,
- When it finishes downloading, close the Library Manager.

Video 01 https://youtu.be/ZMCfNcSNpzM

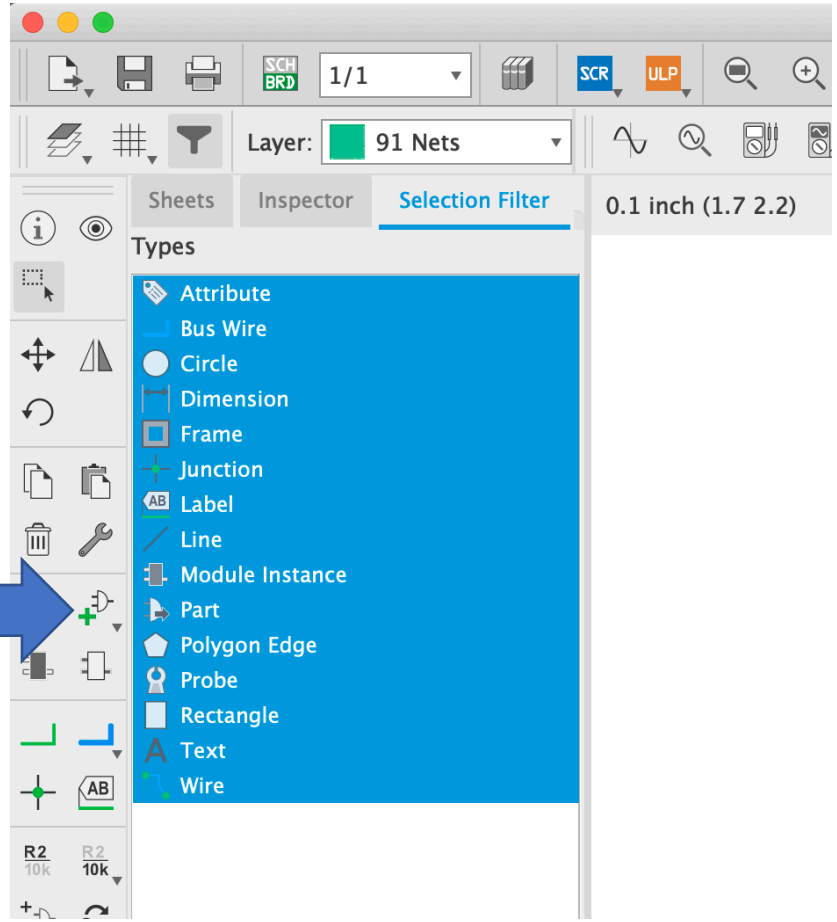


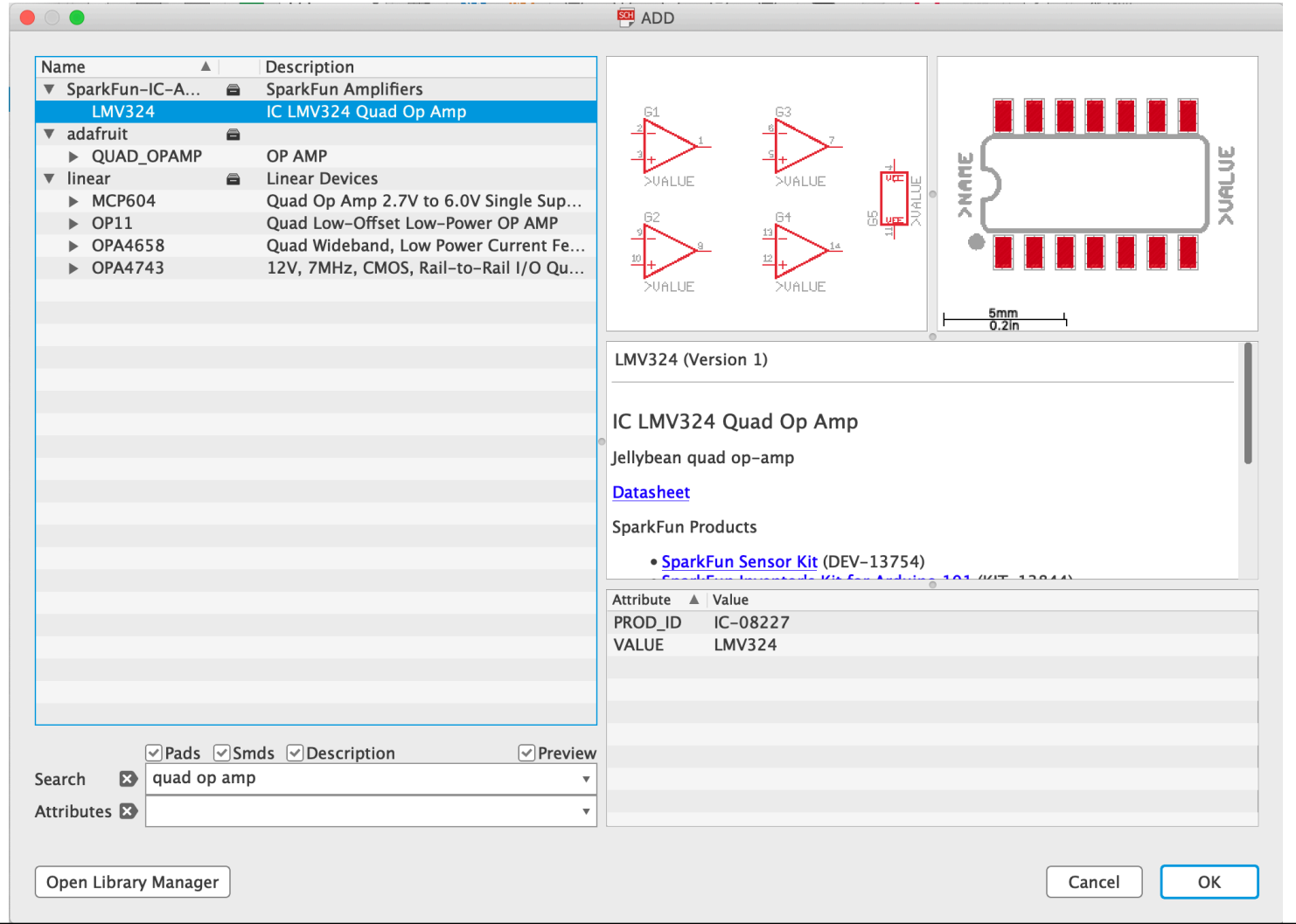
QST February 2019
A Force-Sensing CW Paddle
Art Heft, K8CIT

Errata:
Pins 2 and 3 of U1A and pins 5 and 6 of U1B should be swapped

B1 9v or six AAA cells in series
C1 0.01 µF capacitor
R1, R2 12 kΩ ¼ W
R3, R4 33 kΩ ¼ W
R5, R6 20kΩ trim pot
S1 SPST toggle
U1 op-amp, LM324N
Z1, Z2 1.75” x 1.75” force sensor

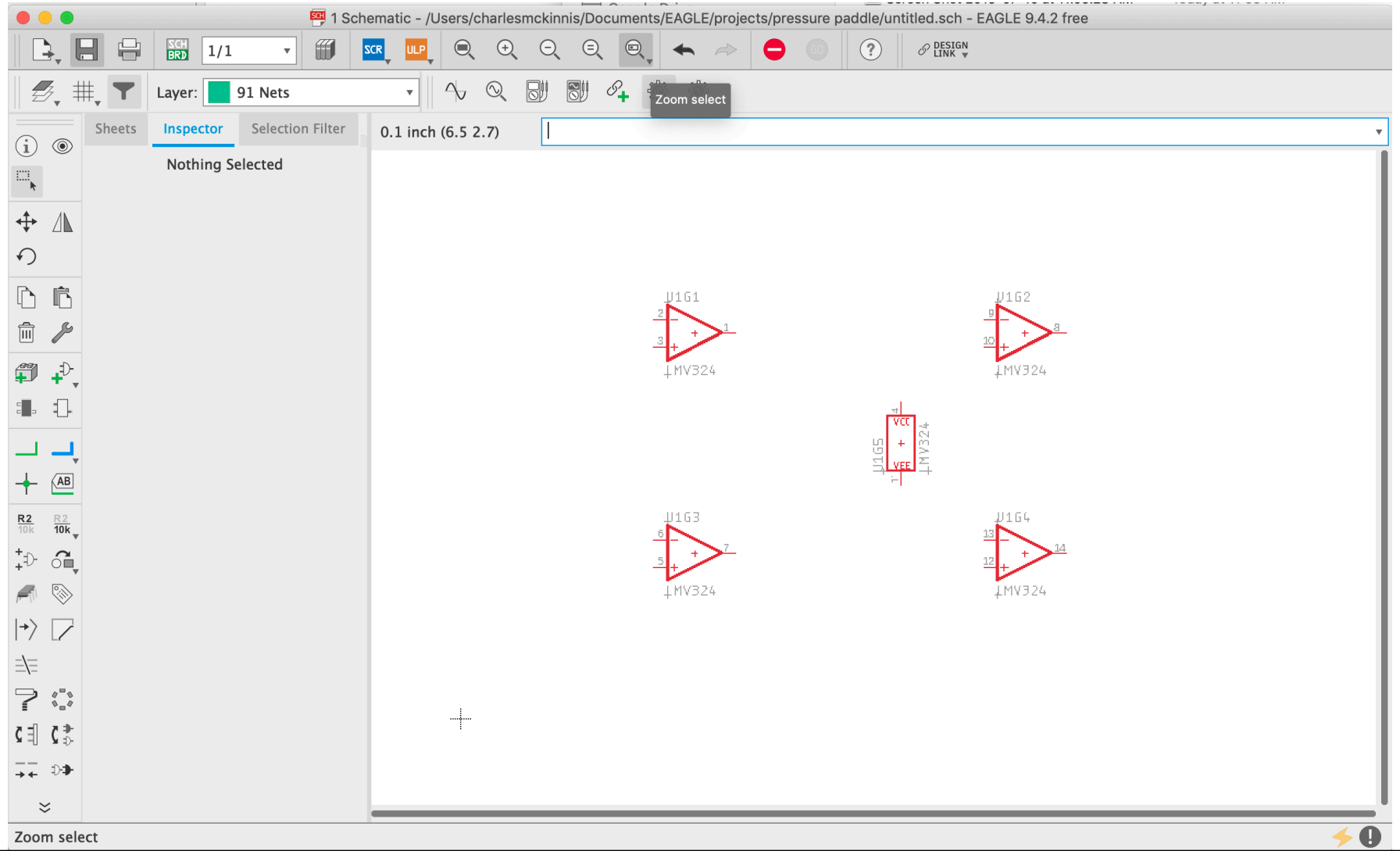
Click the part finder, search for quad op amp and select the device, and click OK. I use the SparkFun listings because they have solder pads with the component.

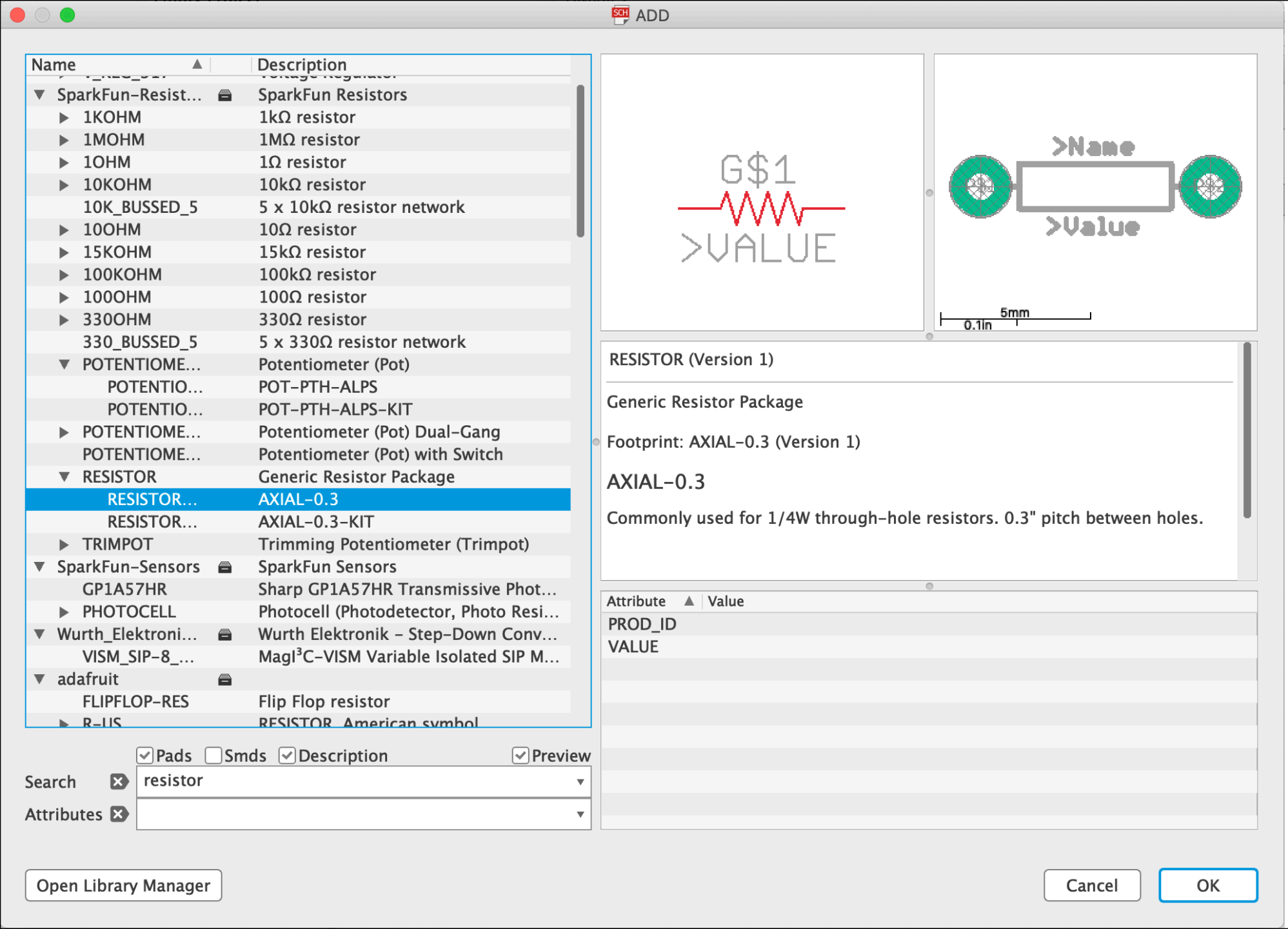




Video 01 https://youtu.be/ZMCfNcSNpzM

Click five times to place each of the four op amps, and the power. Hit esc to return to “Add Part”.





Search for “resistor” and choose the “SparkFun-Resistors” group, scroll to "RESISTOR" with the description “Generic Resistor Package” and select the "RESISTORAXIAL-0.3".

Click OK

Place four on the diagram, and click esc to return to the component library.

More components…

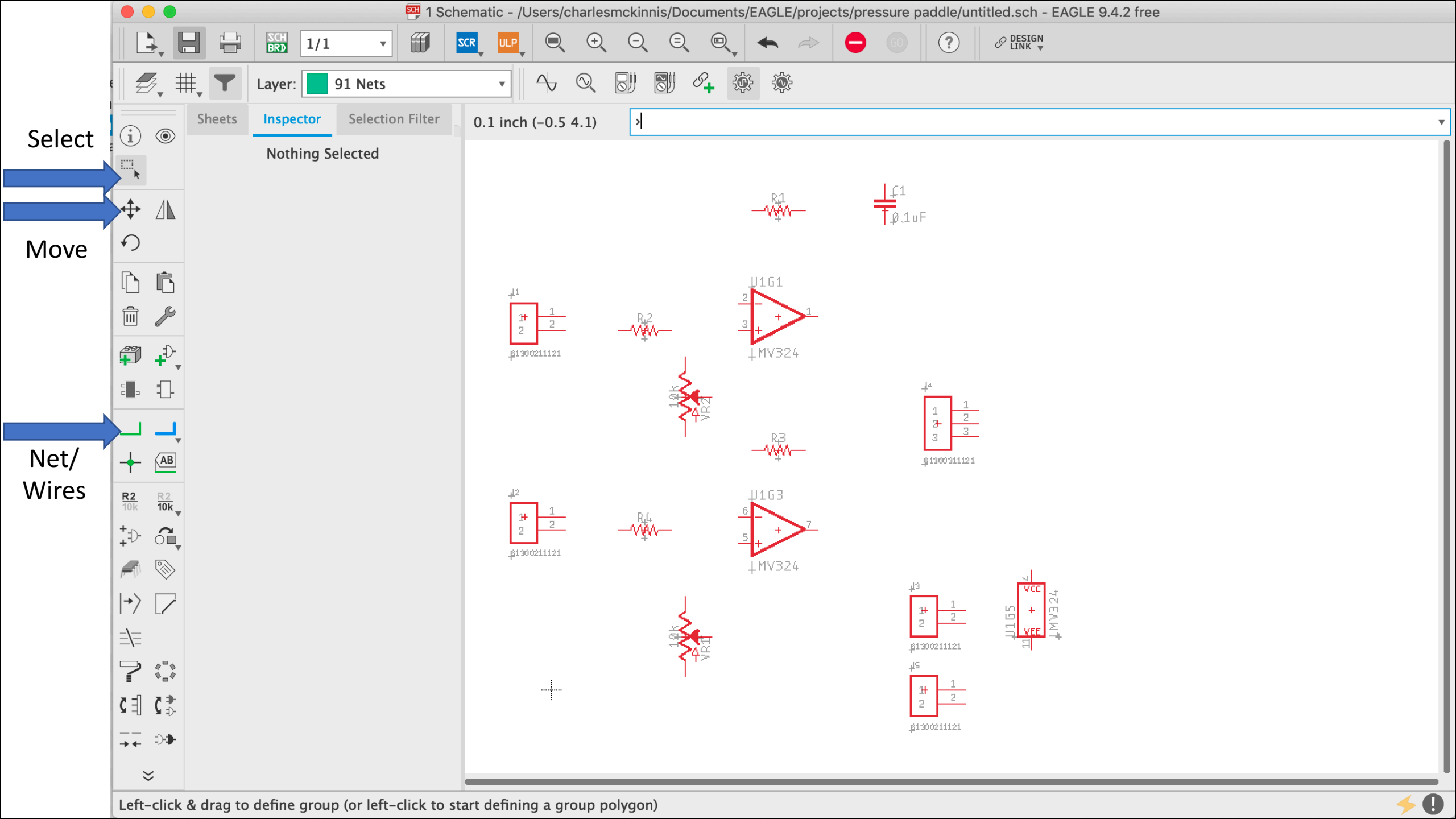
- For the capacitors, search for “capacitor” and select a SparkFun 0.1µF capacitor that is similar in size, description “CAP-PTH-SMALL-KIT”.
- Add one to the schematic, and press esc to return to the component selector.
- Search for “pot” or ”potentiometer”. Again using SparkFun, expanding “Potentiometer (Pot)” and choosing “POT-PTH-ALPS”. Notice in the description this is a “9mm Square Rotary Potentiometer – PTH” and match the part size to what you have or order.
- Place two on the schematic and hit esc to return to the component selector.

Video 01 https://youtu.be/ZMCfNcSNpzM

Headers for sensors, power and output

- Search for “header” and expand the “Headers” category with a description of “PCBLayout.com…”
- Select “CONN HEADER VERT 2POS 2.54MM”; this will create two pads that are 0.1” apart that you can solder wire or headers to.
- Place four headers for the two sensors, the battery connections and an on/off switch. There is a 9v “part” with a layout for a battery holder too, if you prefer. Hit esc.
- Select the 3POS header that is next down on the list, and place one header for the output. Hit esc.
- I added six CONN_01 from the library SparkFun-Connectors to six points. I’ll explain more when we route traces.
- Press cancel to return to the schematic.
- I used a surface mount op amp on the IC so I can demonstrate later.

Video 01 https://youtu.be/ZMCfNcSNpzM


Select
Move
Net/
Wires
1 Schematic - /Users/charlesmckinnis/Documents/EAGLE/projects/pressure paddle/untitled.sch - EAGLE 9.4.2 free
1/1
Layer:
91 Nets
Sheets
Inspector
Selection Filter
Nothing Selected
0.1 inch (-0.5 4.1)
R1
C1
0.1uF
U1G1
MV324
R2
10k
VR2
R3
R4
U1G3
MV324
10k
VR1
U1G5
MV324
VCC
VEE
Left-click & drag to define group (or left-click to start defining a group polygon)

Working with parts

- If you don’t have enough room to move your components, you can zoom in and out by holding ctrl and using the mouse scroll wheel.
- To move a component, click the Move tool on the left, click the component to pick it up, move the mouse and click to put it down where the pointer is.
- To rotate a component, click the Rotate tool on the left, click a component to rotate it 90 degrees counter clockwise per click.
- You can also move the text descriptions around the component with the move tool. Sometimes I click them by accident.
- To rename a component, right click to open the menu and click properties. I changed VR1 and VR2 (Pots) to R5 and R6 to match the schematic.

Video 02 https://youtu.be/tv-_YZfOlMs

Adding a power “bus”

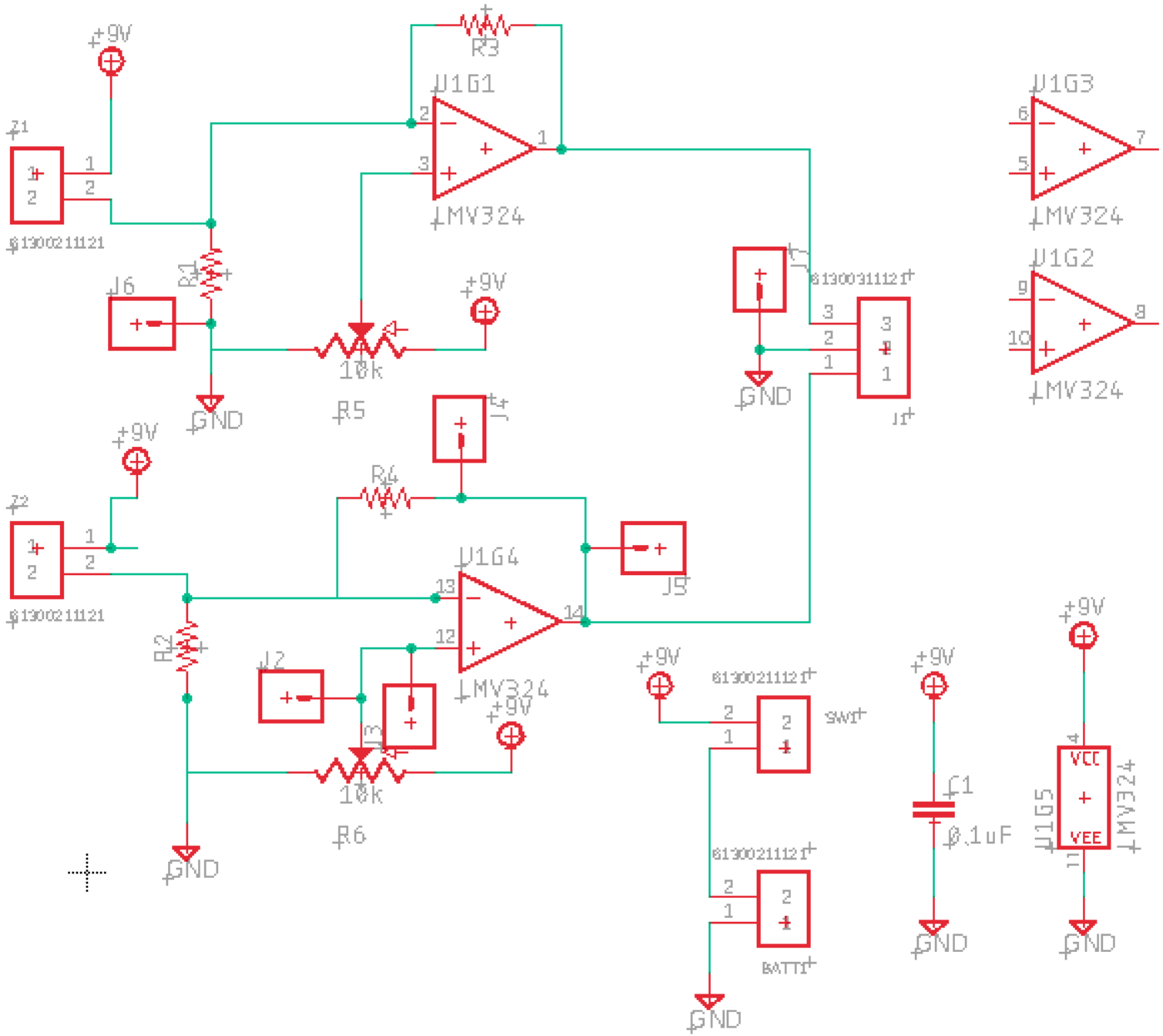
- Add power symbols; Click “Add Part”
- Search for GND, and use SparkFun-PowerSymbols, select GND and click OK.
- Place them in similar places to the schematic. They can be added, for example to the noise filter capacitor too.
- Search for 12v power symbol, and use SparkFun-PowerSymbols, select 12V and click OK. The schematic calls for 9V, but we are using the symbol so it does not matter.
- Place them on the schematic like you did with GND.

Video 02 https://youtu.be/tv-_YZfOlMs

Add “wire”

- On the left toolbar, the green L, on its side, is the Net. This is the wire and will be traces on the circuit board.
- Select the Net, and start connecting components. You can click on the diagram to make a turning point, and esc to end the ”wire”.
- The op amp on the article has pins 2&3 and 5&6 reversed, but the Eagle part has them in the correct location. Connecting the wires where shown made the correct connections.
- I used a different op amp than the schematic to make it easier to route traces. This placed the pads on the opposite side, giving me more room to work on the board.

Video 02 https://youtu.be/tv-_YZfOlMs


+9V
R3
U1G1
MV324
Z1
B1300211121
R1
J6
10k
R5
GND
J7
B1300311121
U1G3
U1G2
J4
R4
Z2
U1G4
J5
R2
J2
J3
R6
B1300211121
SW1
BATT1
C1
0.1uF
U1G5
VCC
VEE

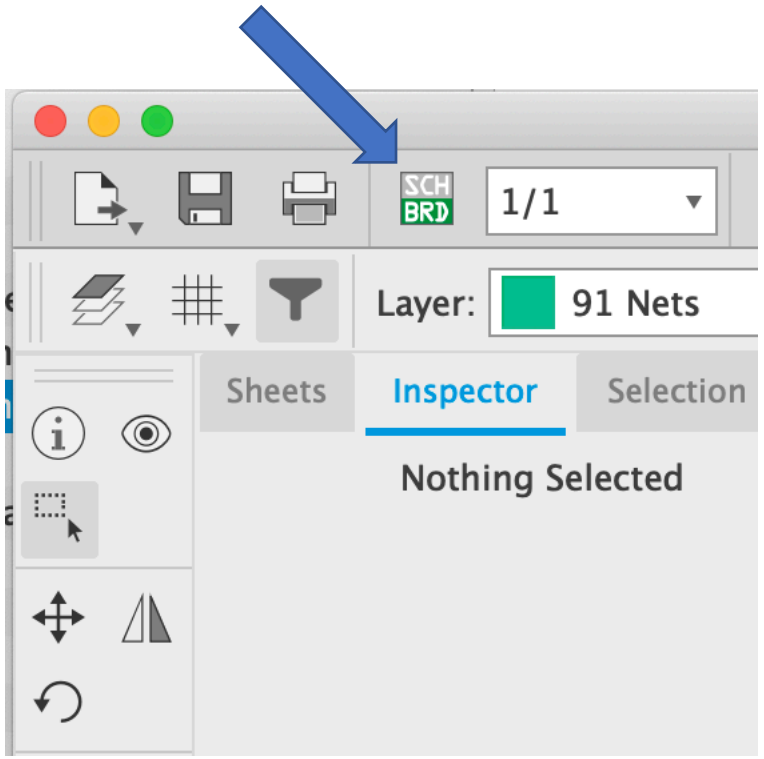
Trace width and clearance

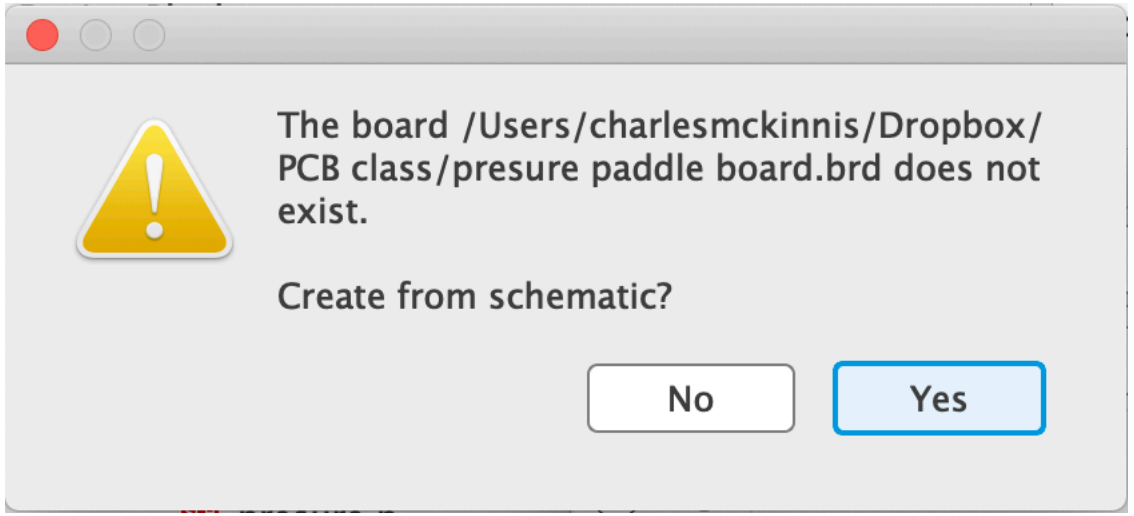
- For laser transfer PCB design, a good starting place for trace width and clearance is 50 mil.
- It didn’t fit the surface mount chip, so I used 25 mil.
- From the top menu click “edit”, “Net classes…”,
- On Nr 0, Name default, change “Width” and “Clearance” to 50mil,
- Click OK.

Video 03 https://youtu.be/Btd9J3dcVA8

Switch to the board design

- Click the “SCH/BRD” button and click Yes to create the board if it asks.





Video 03 https://youtu.be/Btd9J3dcVA8

2 Board - /Users/charlesmckinnis/Dropbox/PCB class/presure paddle board.brd - EAGLE 9.4.2 free

Layer: 1 Top

Design Manager | Inspector | Selection Filter

Devices | Signals

Device Sets — 3 of 3 shown (1 selected)

Search

Device Set
<All Devices>
<Bottom Side Devices>
<Top Side Devices>

Devices — 13 of 13 shown (0 selected)

Search

Name	Footprint	Value	Side	F
BATT1	61300211121	61300211121	TOP	
C1	CAP-PTH-SMALL-KIT	0.1uF	TOP	
KEYER	61300311121	61300311121	TOP	
R1	AXIAL-0.3		TOP	
R2	AXIAL-0.3		TOP	
R3	AXIAL-0.3		TOP	
R4	AXIAL-0.3		TOP	
R5	POT-PTH-ALPS	10k	TOP	

Items — 0 of 0 shown (0 selected)

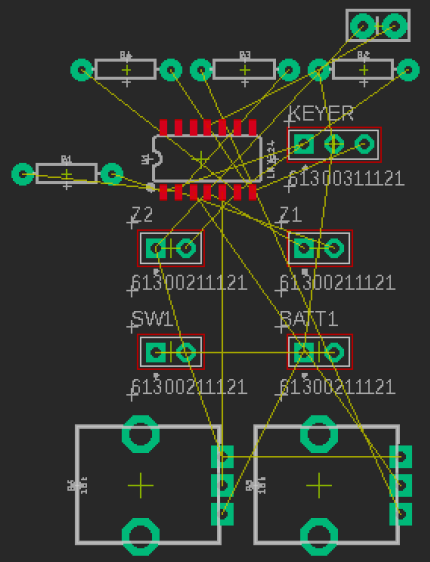
Search

Type	Name	Signal	Layer	Size (mil)	Pos. (mil)	Angle

50 mil (-290 -433)

Well, this is ugly.

The components



The black box, with the yellow line represents the board.

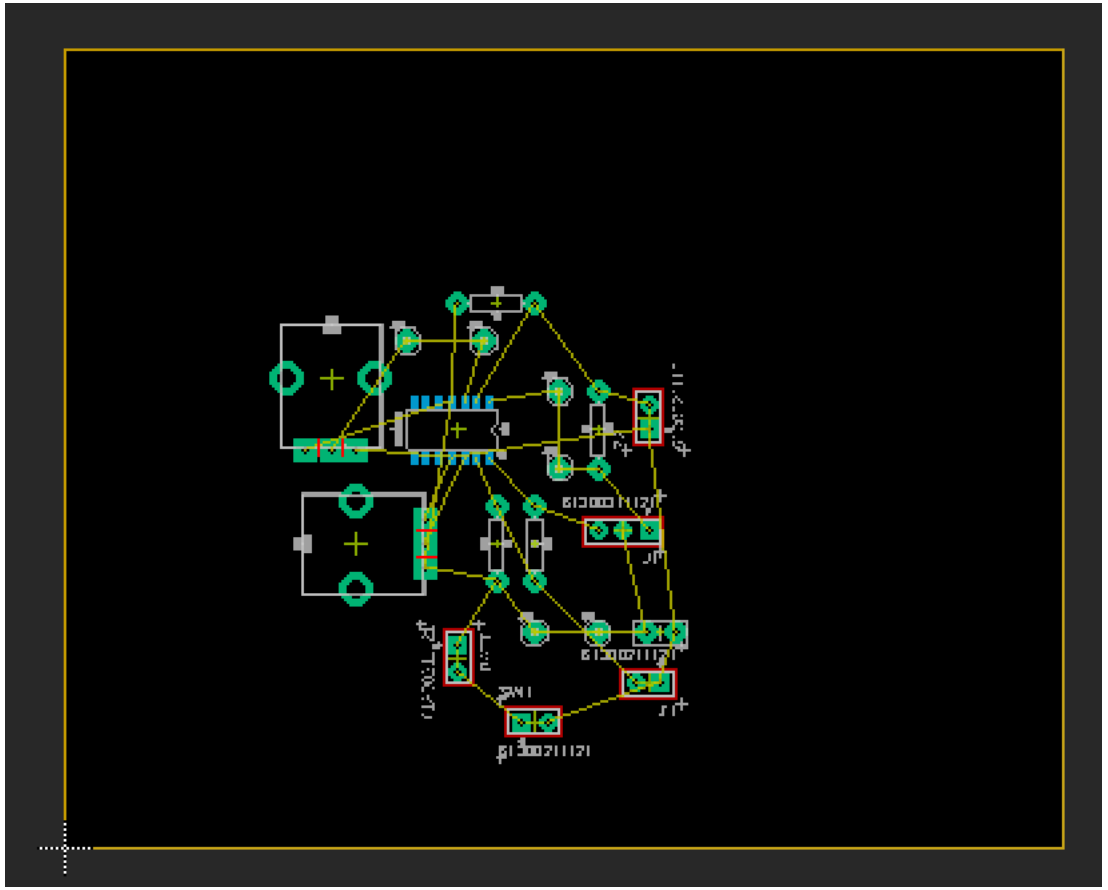
MANUFACTURING

FUSION 360

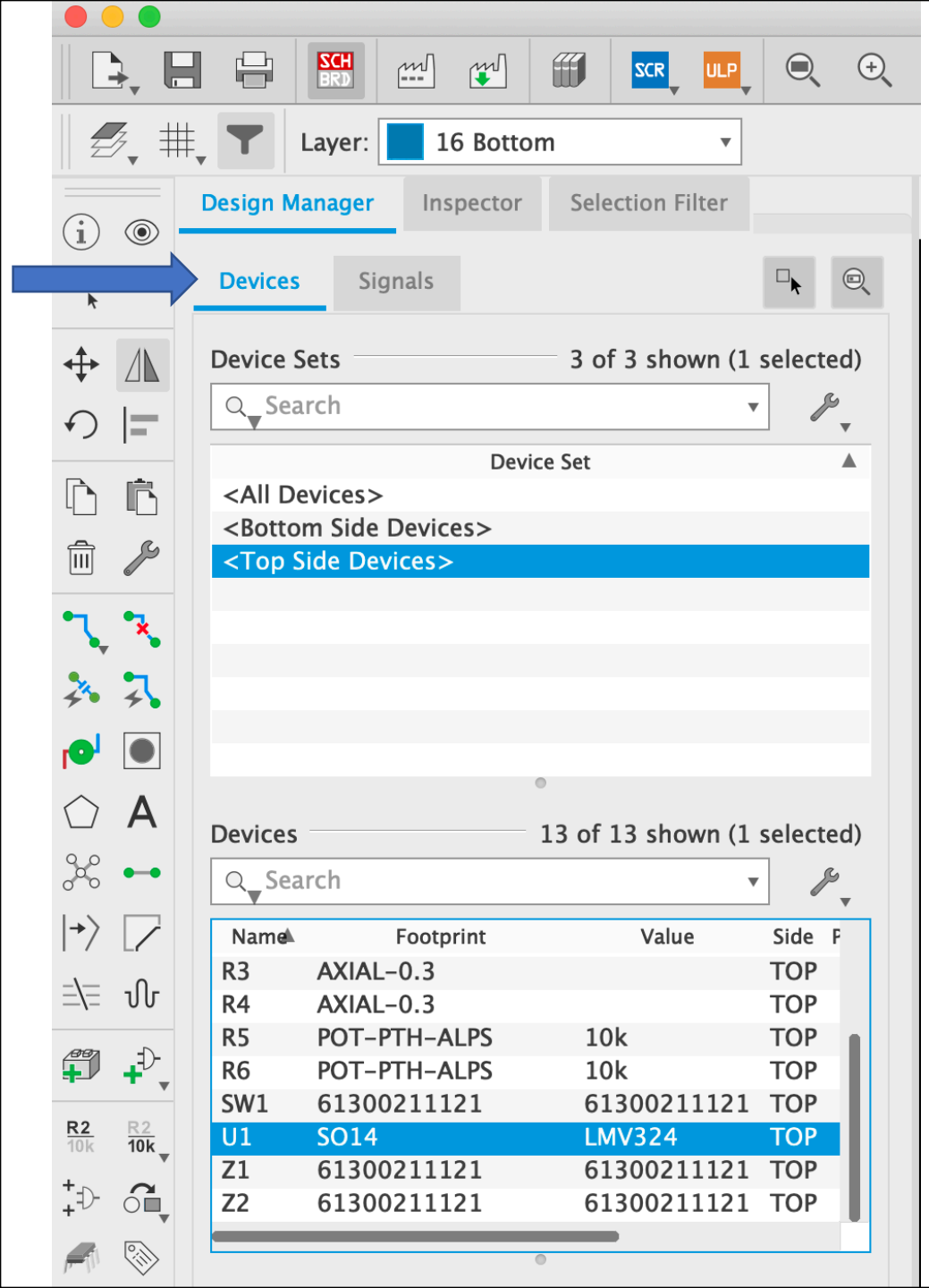
FUSION TEAM

Well, that’s ugly.

- Select all the components, and drag them to the board (the black box with the yellow line).



Video 03 https://youtu.be/Btd9J3dcVA8



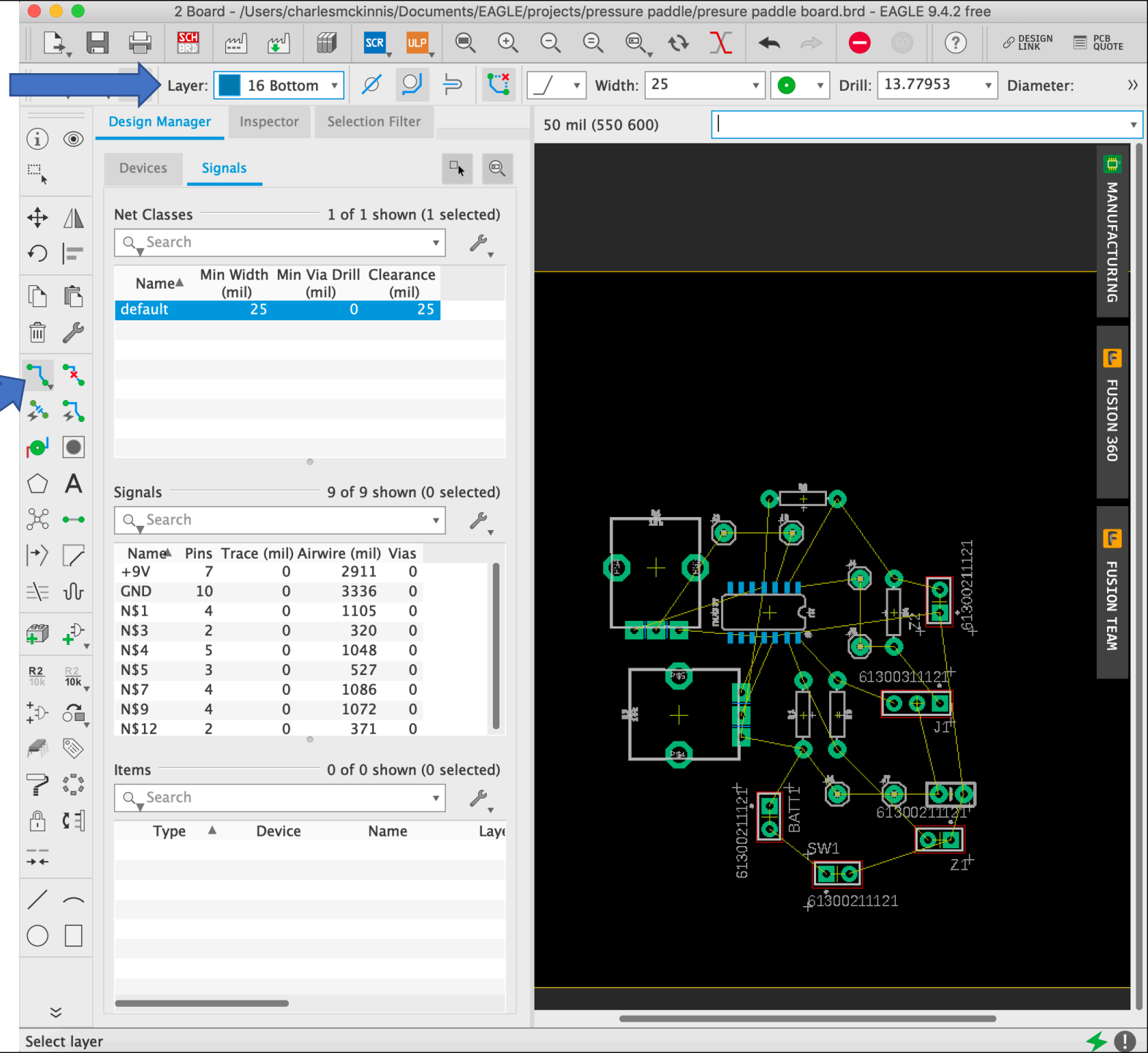
Surface Mount Chip

- The IC is a surface mount chip. The board only has copper on one, so we need it to be on the “bottom”.
- Select the Device Manager tab on the left, and the Devices tab below it,
- Find the chip in the bottom windows and double-click it,
- In the pop-up window, checkmark “Mirror” and click OK.
- It is just a checkmark box, so there is no picture.

video-PCB-003

Bottom layer and traces

- At the top, choose the bottom layer, where our traces will go.
- On the left, choose the “Route Airwire”.



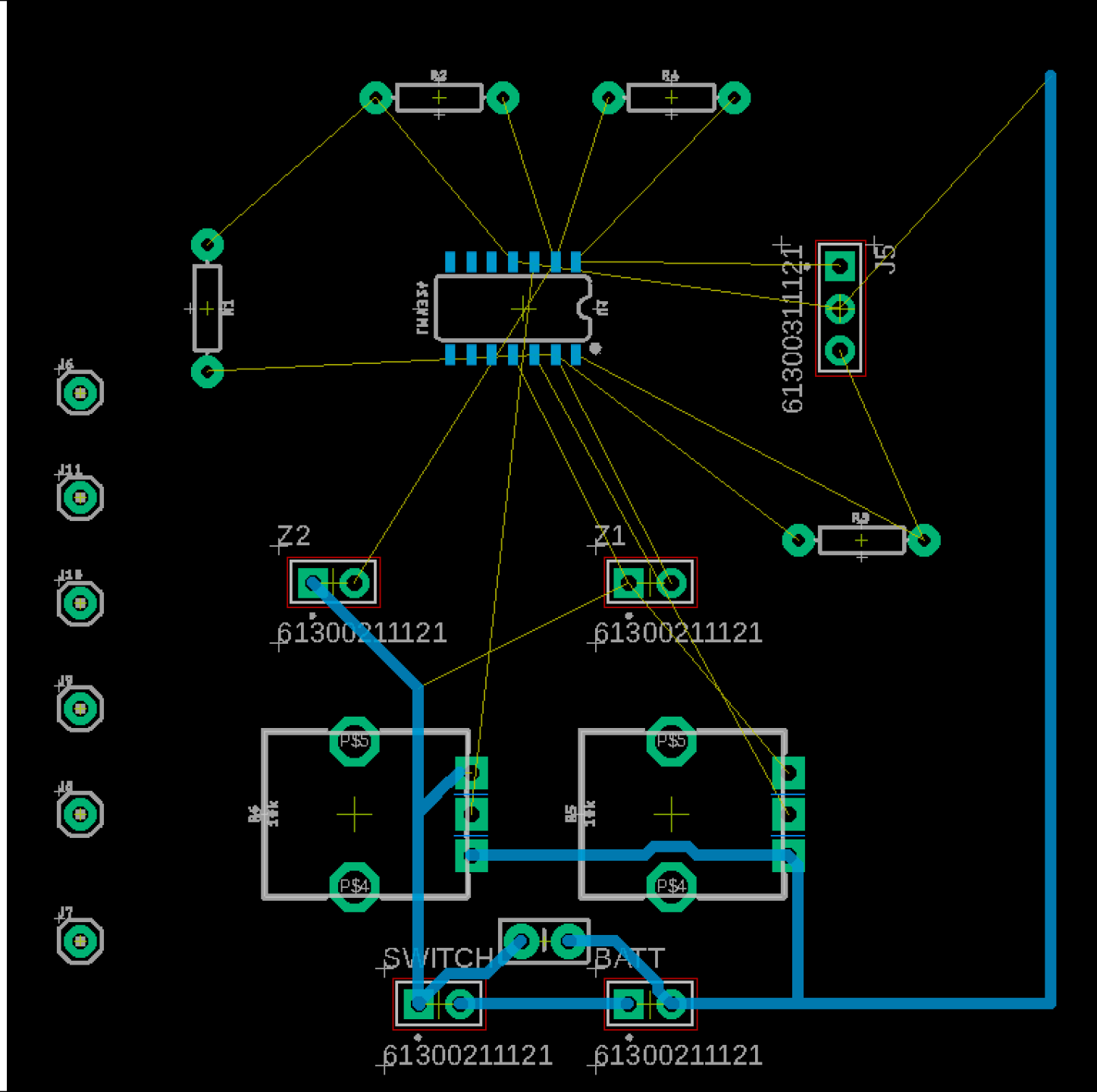
Video 03 https://youtu.be/Btd9J3dcVA8

Order from Chaos

- The IC pins will be the most crowded, so pull components outward.
- Much of the arrangement will be orienting parts and untangling the “airwires”.
- Because traces can’t cross, we can pass traces between the resistor pads and use them as jumpers.

Bottom layer and traces

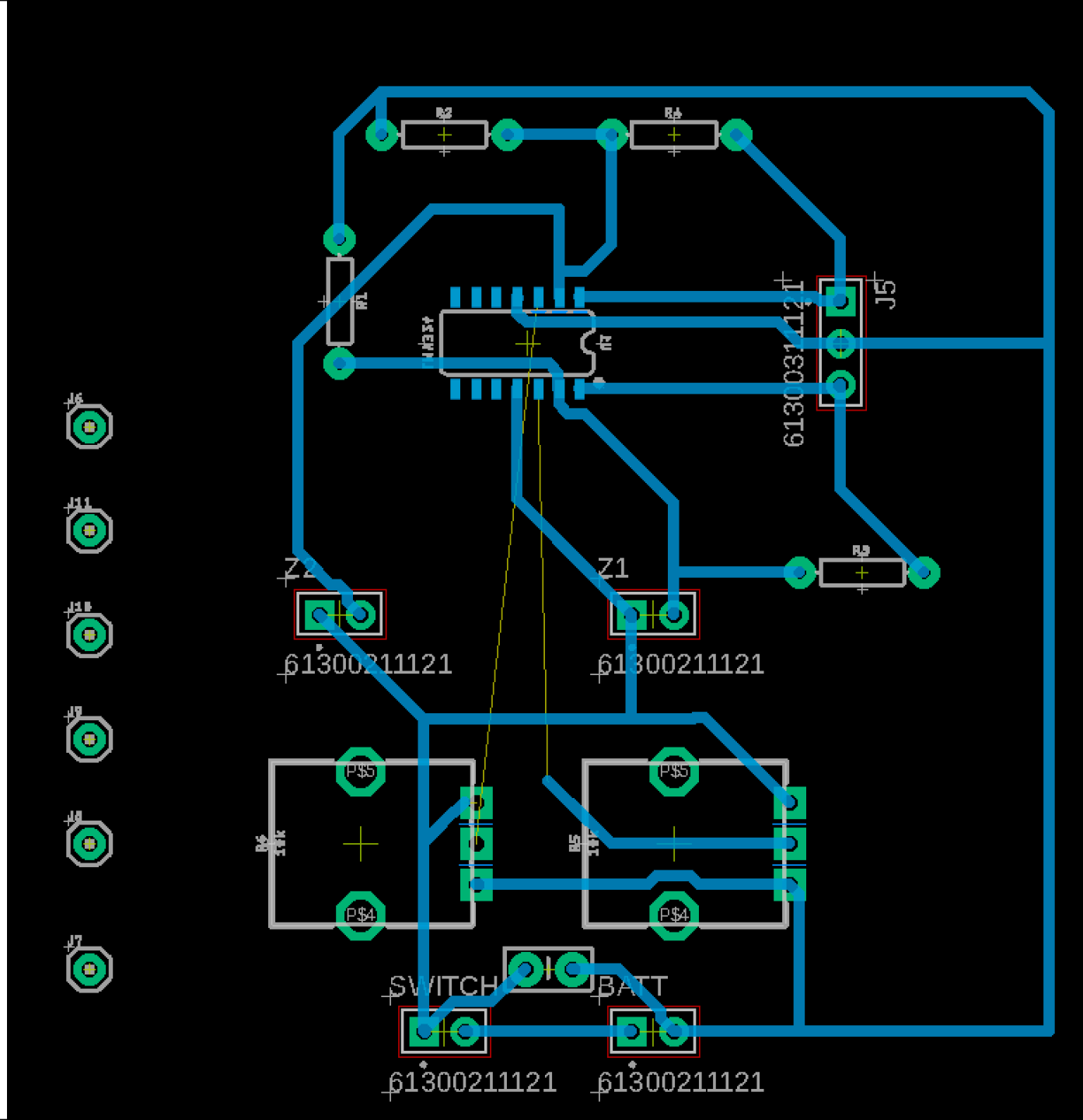
- Connect the switch to the battery, connect the filter capacitor,
- Move the CONN_01 (positive) pads to the left, and start with a ground trace on the right.



Video 04 https://youtu.be/TpE1hM2i43A

Bottom layer and traces

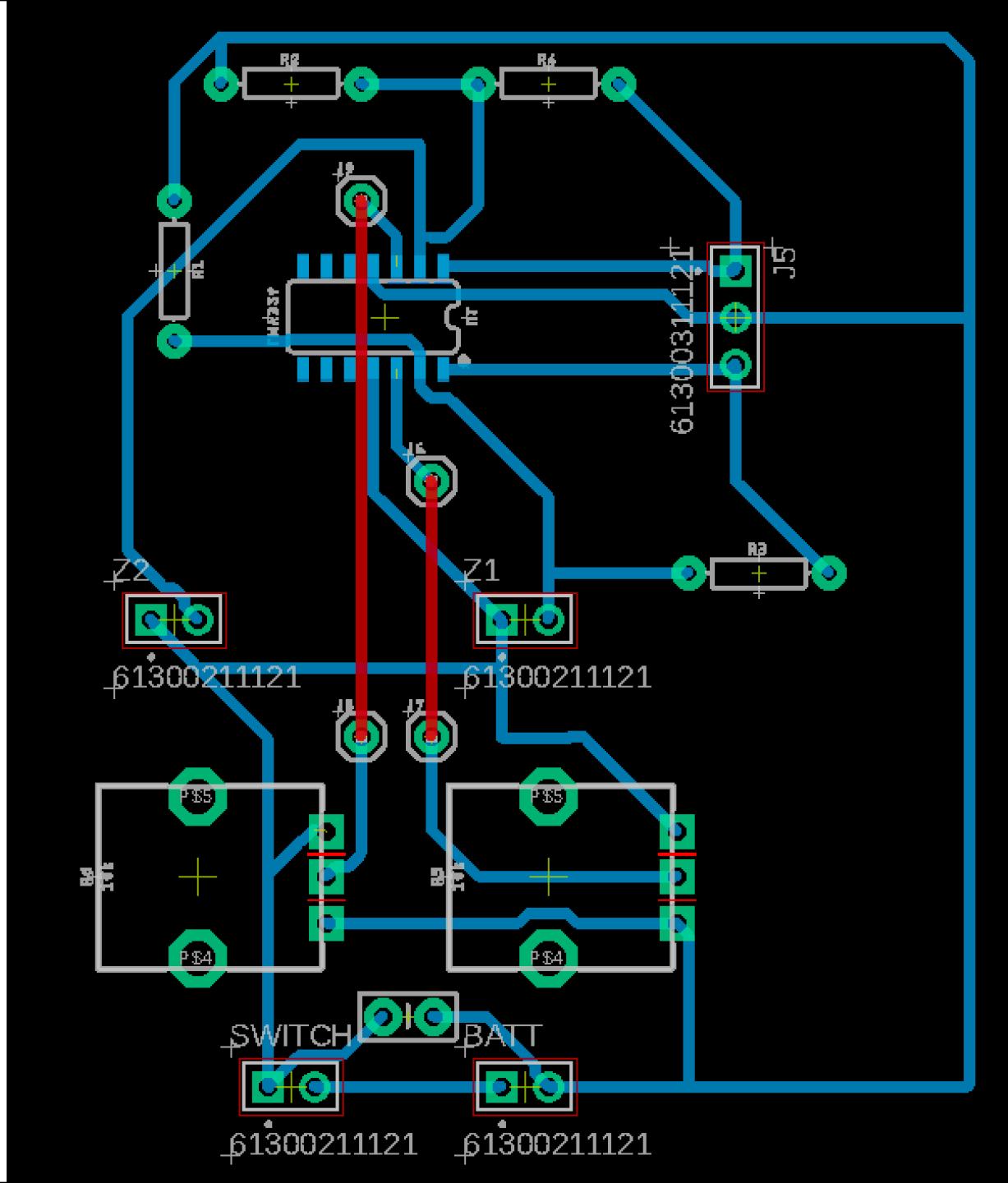
- Because traces can’t cross, we can pass traces between the resistor pads and use them as jumpers.
- A couple of air wires cross other traces…



Video 05 https://youtu.be/KOWHothoJn4

Un-connections

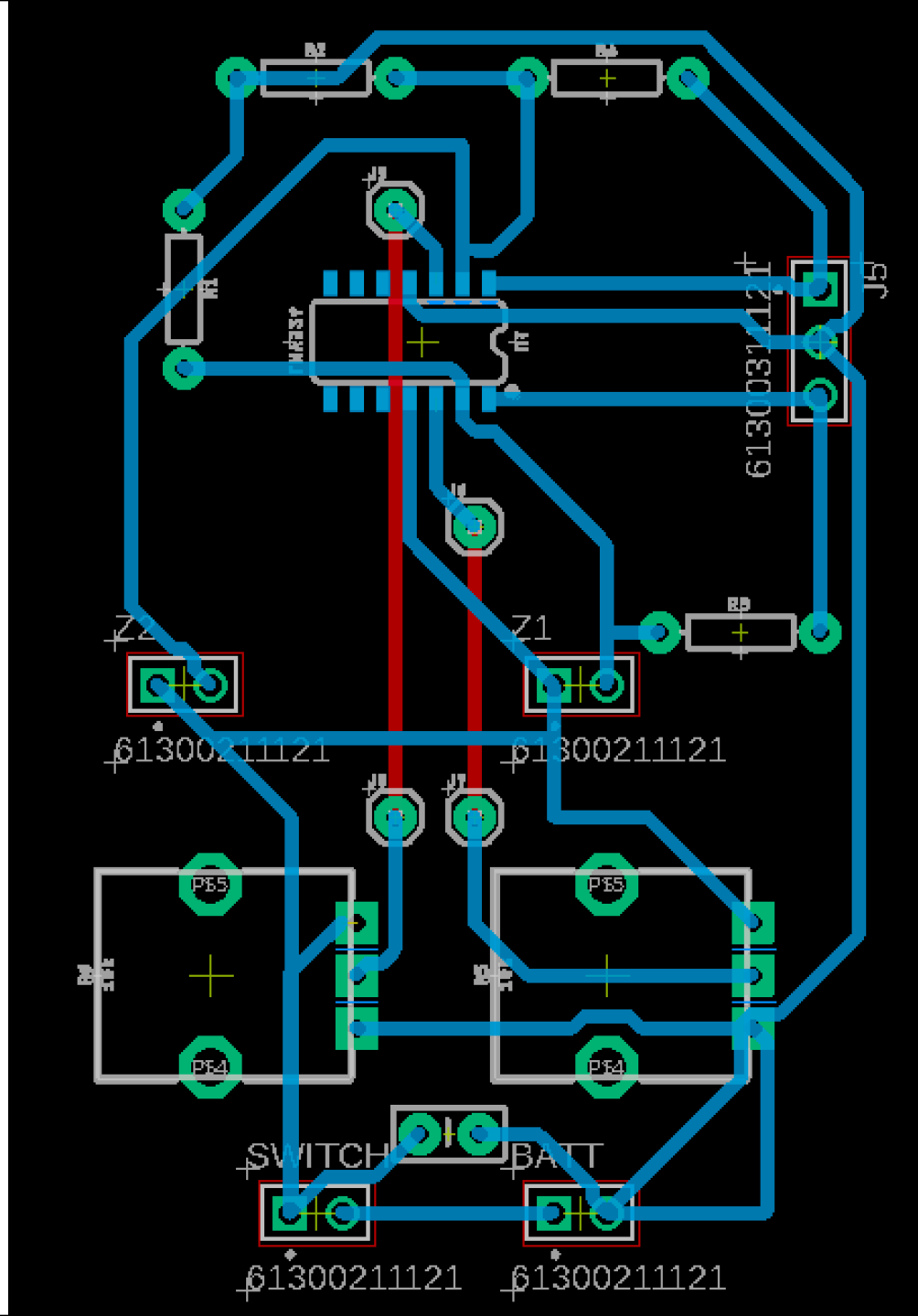
- A couple of air wires cross other traces.
- These are the pot middle post to the IC.
- We will attach a CONN_01 to both sides of that connection on the schematic.



Video 06 https://youtu.be/uSeZPp7hLLI

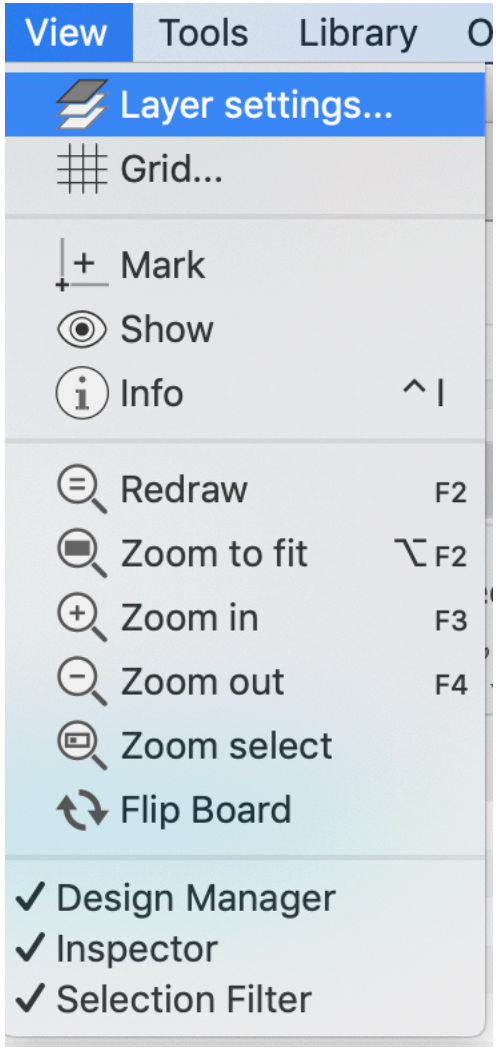
Clean-up

- Move components in,
- Re-run traces that can be more efficient.



Video 07 https://youtu.be/pIpTfJMGY2o

Prepare to print!

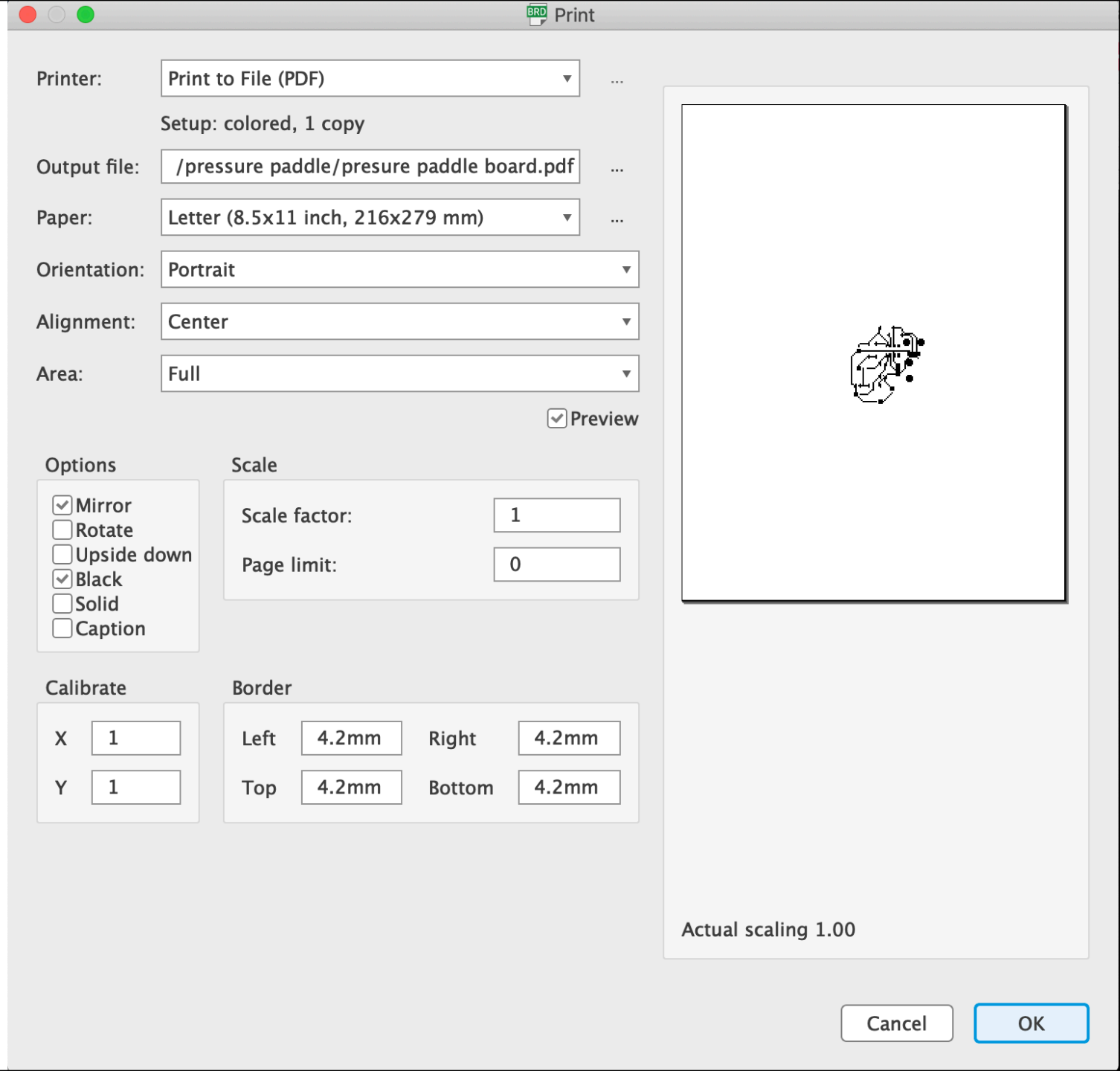


Click the “View” menu at the top, Layer settings...

Click the Hide Layers button. Then tap to the right of Bottom and Pads to make them the only traces visible.

Print!

- Choose “Mirror”
- Choose “Black”
- Print on a magazine page.



Prepare the PCB

- Use a green kitchen scrubber, sand paper or steel wool to scrub the copper surface.
- This removes corrosion and provides a more textured surface for the toner to adhere to.
- Clean the copper with pure alcohol (not rubbing alcohol from the pharmacy department).
- Blue paper towels, commonly found in the automotive department work great because they have less fuzz, and are more durable than other varieties of paper towels.

Videos of the etching steps are available on the YouTube.com KU5N Channel
https://www.youtube.com/playlist?list=PLIvKeVmi3PzTqlgM-J9iKa25MHVGVr15-

Transfer the printer image to the PCB, with a clothes iron

- High heat (Wool is a good setting),
- A solid surface under the board that will tolerate heat (3/4” MDF board works),
- Cut the pattern out, with a small margin,
- Align the design with the toner facing the board,
- Set the iron over the pattern for about 1 minute and press down,
- Move the iron around the edges, leaving it in each place for about 15 seconds, each move should overlap with the last, making sure the center and whole outside edge gets heat and pressure.
- CAUTION! The board will be hot.
- Let the board cool.

Video https://youtu.be/iWsjbD_YI0k

Etching Equipment

- Eye protection,
- Nitrile gloves, and
- Wear a long sleeve shirt.
- A plastic tray about 18” x 24”, that will hold an inch of water. This will be the splash tray.
- A small plastic container with a secure lid that is narrow enough that the board will not lay flat in the bottom; 45 degrees is a good angle. This will be the etching tank.
- Muriatic Acid (found at most hardware stores),
- Hydrogen Peroxide (3%, found in the pharmacy department), and a
- Fish tank bubbler and clear tubing.

Etching Safety First!

- Wear eye protection,
- Wear nitrile gloves, and
- Wear a long sleeve shirt.
- If the muriatic acid splashes, flush the area with water.
- Only use muriatic acid in well ventilated areas. Be careful while handling, as it will cause chemical burns.

Prepare the etching equipment

- Put the splash tray on a stable surface, and add an inch of water. This is to dilute the Muriatic Acid if it splashes.
- The small container is the etching tank. The quantity will vary depending on the size of container, so we will use a ratio. You want the board to be completely submerged.
- Do not place the board in yet. If you are not sure about the liquid quantity, measure the amount of water needed to cover the board.
- Add 2 parts Hydrogen Peroxide to the etching tank,
- Add 1 part Muriatic Acid (sold at hardware stores in the concrete section) to the etching tank.

Video prep https://youtu.be/oCpeHW_fF9g
Etching https://youtu.be/51vcpg-s4ik

Etching the board

- Plug in the fish bubbler, and place the end of the tube in the etching tank.
- Place the copper clad board, with the copper facing down, and the bubble running up the surface.
- As the copper is removed from the board, you can carefully move the air tube to other areas to accelerate those.
- When only the covered traces remain, remove the tube from the tank and submerge the end in the water tray to rinse it.
- Remove the board carefully with gloved hands. Submerge the board and your fingers in the water to rinse them.
- Place a secure lid on the etching tank.

Acetone to remove toner

- Acetone is used to remove the toner.
- Nail polish remover can be found as pure acetone (check the contents), and is less expensive than a gallon can from the hardware store.
- Blue paper towels, commonly found in the automotive department work great because they have less fuzz, and are more durable than other varieties of paper towels.
- Apply some acetone to a paper towel, and wipe the toner from the surface.

Drill holes

- Find the appropriate size drill bits for your component leads, and drill holes in the pads.
- Micro drill bits are available online. They break easily, so you either want to get a range, or a collection of common sizes.
- The Dremel Workstation functions like a drill press for Dremel rotary tools. Because it only moves up and down it is easy to position the board, slowly lower the bit to check position, drill the hole, and lift with very little risk of breaking the drill bit.
- A pin vise or small spiral manual push drill work too.
- I made holes at the edge of an extra board, and wrote the size next to each. Then used the component leads in those holes to determine which size to use.

Video https://youtu.be/14Egchp9Ces

A little about solder

- The most common electronics solder is 60/40, 60% tin (Sn) and 40% lead (Pb). It has a liquid temperature of 370F (188C).
- 60/40 solder has a plastic state, between solid and liquid. When a component moves while cooling, it causes a frosted glass appearance and a weaker solder joint.
- For about the same price, you can buy 63/37 “eutectic” solder. Eutectic solder does not have a plastic state, it goes from liquid to solid and will not create a frosted solder joint. The melting point is 361F (183C). Go buy some and thank me later.
- Solder paste is available in these blends, and is a paste rather than a strand. You apply the paste to the pads on the board, place the components and they will stay in place, then heat the paste to melt the solder.

Surface Mount components first

- It is normally not needed, but you can mask a trace with enamel paint to prevent the solder from running. There are enamel paint pens that work well for masking the traces. Avoid acrylic paint pens; they will not work.
- Apply solder paste to the pads; only a little bit is needed to cover each pad.
- Place the IC chip on the pads, and the paste will keep the chip in the right general area.
- To melt the solder paste, use a butane iron with a hot air tip, or a kitchen hot plate.
- As the solder melts, the solder will move off the board and to the copper trace or pad, the surface tension will break and the chip will settle and align to the pads.
- You can use a dental pick to nudge it, if needed.
- When the solder liquifies, remove the heat. For a hot plate, use a spatula to lift the board out.
- Safety first: Dedicate a hot plate and spatula to solder. Do not mix food use and solder use.

Solder the through hole components

- For axial components, like the resisters, bend the leads and slip them through the holes. The leads should protrude from the side with the copper traces.
- Bending the leads out slightly on the bottom will keep them from falling out.
- Put a little solder on the iron tip, place it in contact with the lead and pad, and add a little solder. The amount of solder should form a triangle with the top slightly caved in.
- Sliding the iron up the lead will remove the heat and avoid “tails”.
- Clip the leads with a flush cutter just above the solder. Touching the joint with the iron again will reflow the joint, and can make it look nicer.

Cleaning the board

- Spray the board with alcohol and brush it with a hog hair brush, or the free toothbrush the dentist gave you to remove flux and make those joints shine.
- A small atomizer bottle from the travel department works well as an alcohol sprayer.
- Don’t use your spouse’s toothbrush, they will find out and won’t be happy.

Celebrate, and present to the Ham Club!