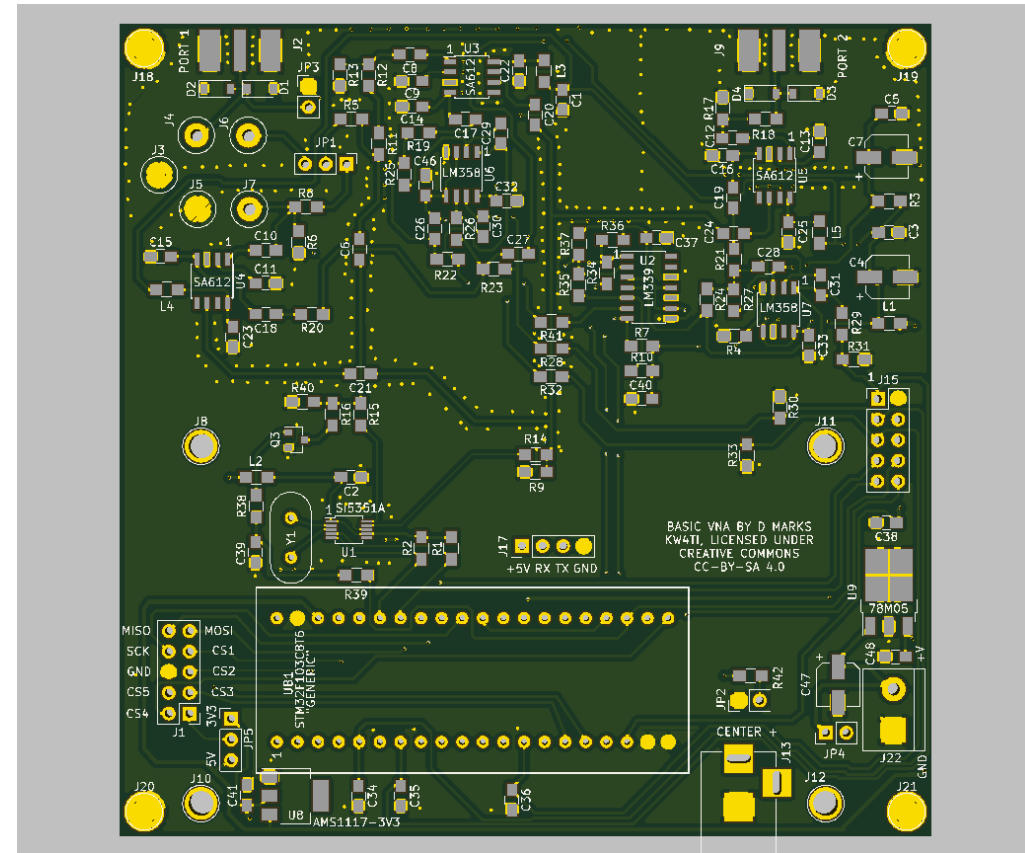
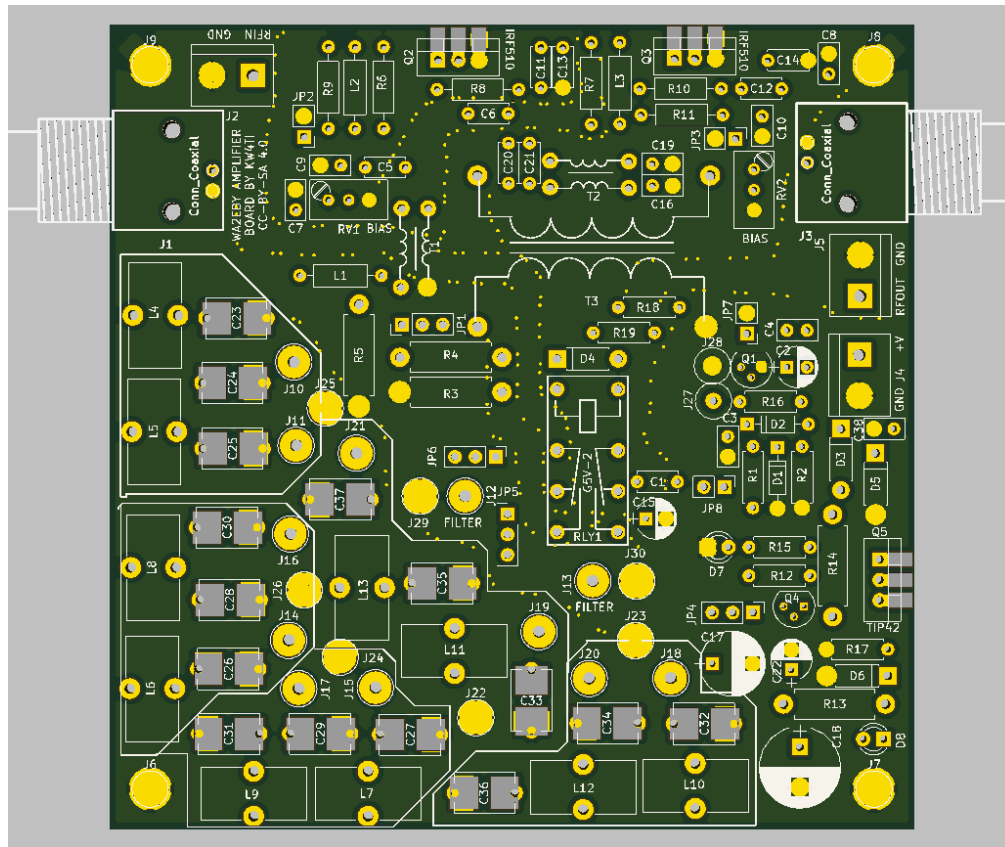


Open Source for Amateur Radio Projects Including Kicad PCB design

By Daniel Marks, KW4TI

Raleigh Amateur Radio Society (rars.org)

September 10, 2019



What is “Open Source”?

It is a way to license software so that it remains “free.”



Free as in “beer,” you get free beer, you drink free beer, you don’t pay for it. Isn’t that great?

Free as in “libre,” the rights you receive as a software user, and guarantees as a developer.

- As a developer, your contributions to a free software project stay free (not made proprietary).
- As a user, your rights to use the software are irrevocable if you abide by the license.
- The source code is provided.
- As a developer, your software is provided with no warranty, and you are not liable for any damages.

The licenses help developers and users cooperate.

Open Source in amateur radio:

It makes software and hardware tools available that help and encourage participation and enjoyment of the hobby.

(Highly incomplete list) examples of software:

- FI-digi: amateur digital modes, (PSK31, etc.)
- WSJT-X (FT8, JT65, etc.)
- LOG4OM, cqrlog (logging software).
- Dire Wolf terminal node controller software
- CHIRP (program your transceiver).
- NEC2 (antenna field solver) – 4NEC is non open source but free front-end
- Kicad (PCB design software).
- Qucs (SPICE-based graphical circuit simulator)

Open Source Hardware in amateur radio:

Briefly, this means freely available schematics, PCBs, and firmware for hardware devices. Examples:

- uBitx/BITX 40 QRP transceiver
- mCHF transceiver
- EU1KY Antenna Analyzer
- W8TEE Arduino Antenna Analyzer
- Haasoscope USB digital oscilloscope
- HPSSDR (High Performance Software Defined Radio) project (<https://openhpsdr.org/>)
- HackRF One software defined radio
- LimeSDR software defined radio

My own projects I will present today....

Developing your own ideas!

Subpart A—General Provisions

§97.1 Basis and purpose.

The rules and regulations in this part are designed to provide an amateur radio service having a fundamental purpose as expressed in the following principles:

(a) Recognition and enhancement of the value of the amateur service to the public as a voluntary noncommercial communication service, particularly with respect to providing emergency communications.

(b) Continuation and extension of the amateur's proven ability to contribute to the advancement of the radio art.

(c) Encouragement and improvement of the amateur service through rules which provide for advancing skills in both the communication and technical phases of the art.

(d) Expansion of the existing reservoir within the amateur radio service of trained operators, technicians, and electronics experts.

(e) Continuation and extension of the amateur's unique ability to enhance international goodwill.

The tools of hardware and software development have never been as easy to use, cheaper, or more available as they are today. There are resources today that previous generations of hams could only dream about.

Did the Internet kill ham radio?

Quite the opposite. It provides the means for hams across the world to collaborate on projects. The benefits of these projects can be shared by all because the rights to the work are guaranteed by

Open Source Licenses.

But 21st century ham radio is going to be very different.
Examples:

Complex digital modulation. (e.g. FT8/JT65)

Smart Antennas. (e.g. phased array antennas)

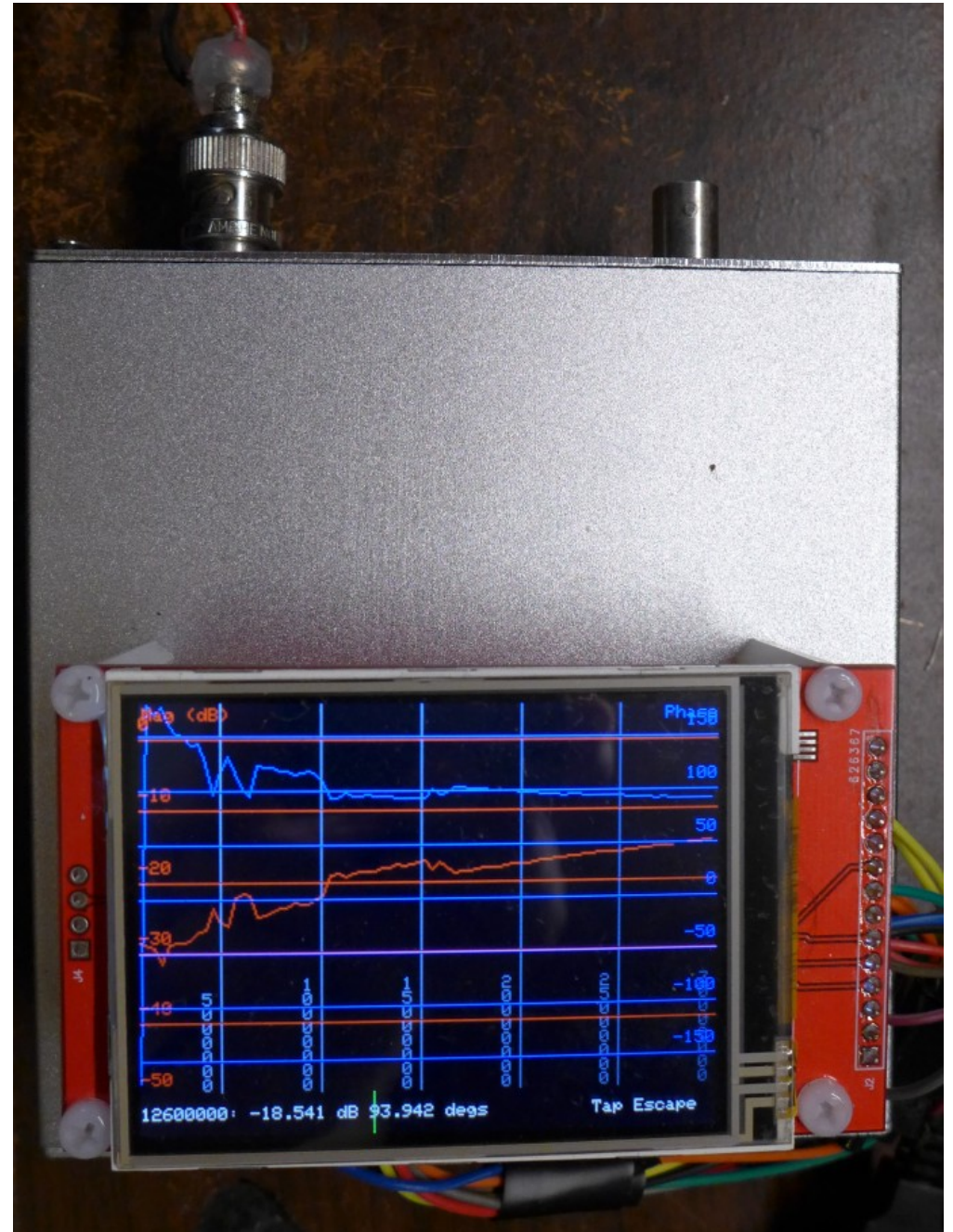
MIMO. Multiple input, multiple output antenna arrays.

Examples from my open source projects: a Vector Network Analyzer

A VNA is a device for measuring the reflections and transmissions from radio frequency components, for example, antennas and filters.

It can be used as an antenna analyzer, or to check the impedance of a balun, or the transmission of diplexer.

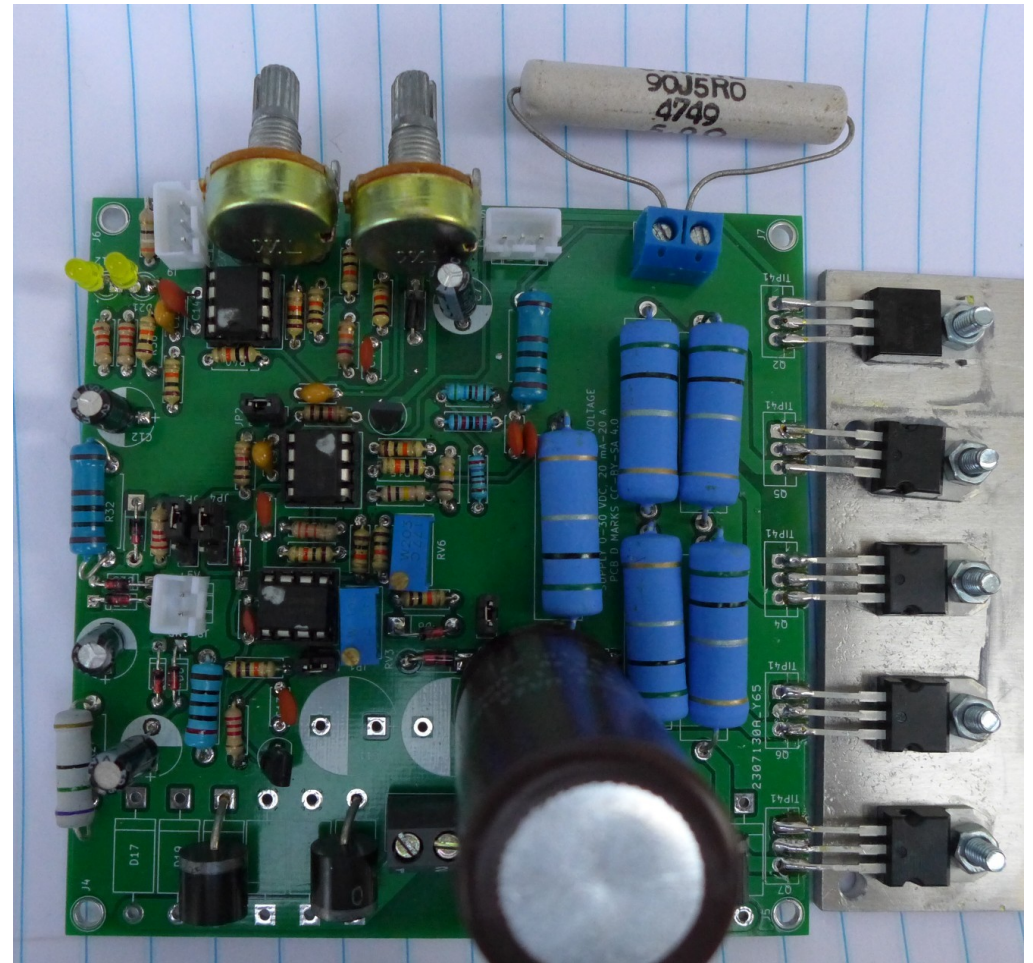
Can be built for under \$50 in parts!



Examples from my open source projects: a linear power supply

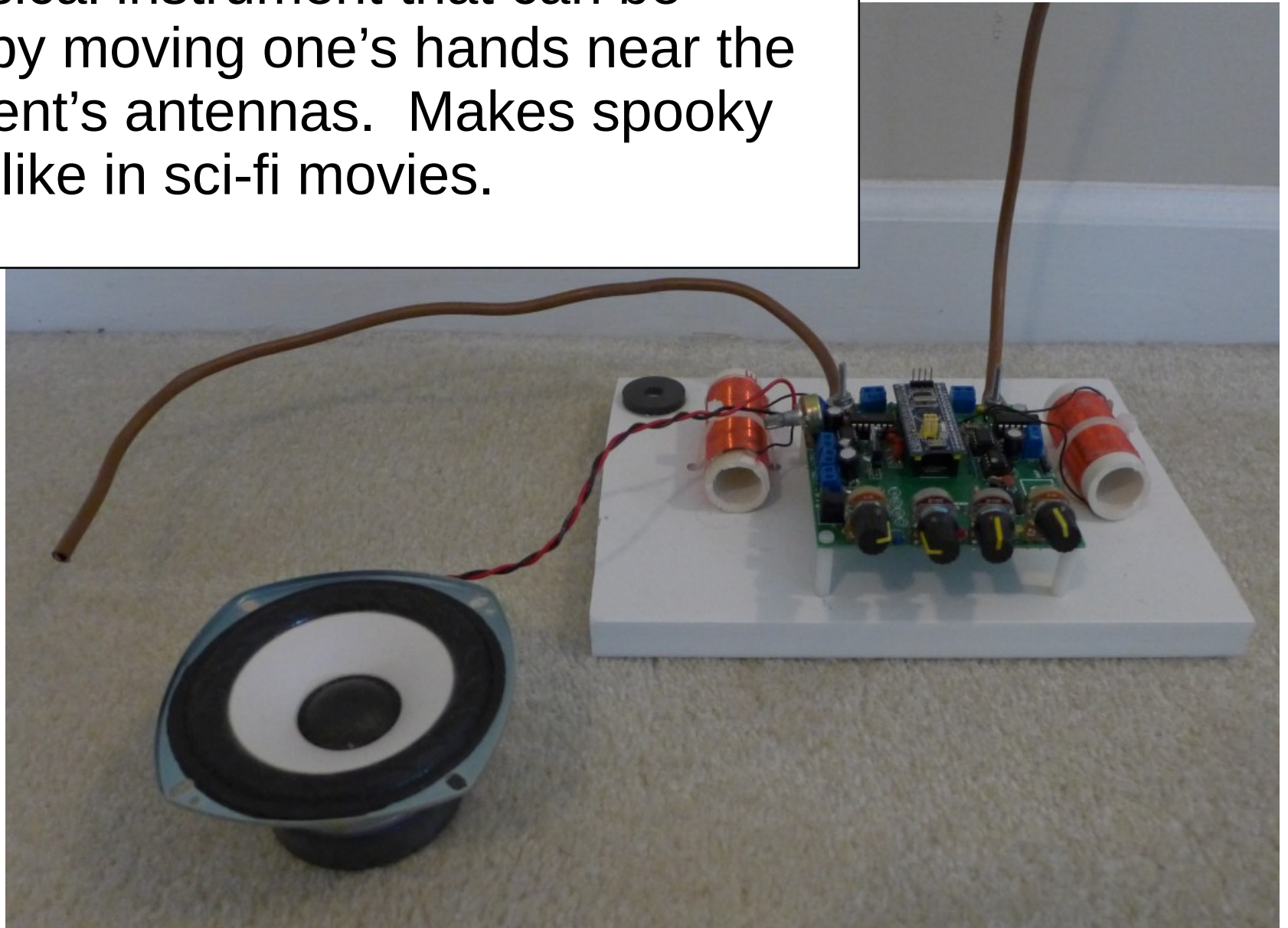
This power supply allows one to make a new benchtop power supply from old, broken power linear power supply parts such as transformers, capacitors, and pass transistors.

This power supply has constant voltage and current capability and external control.



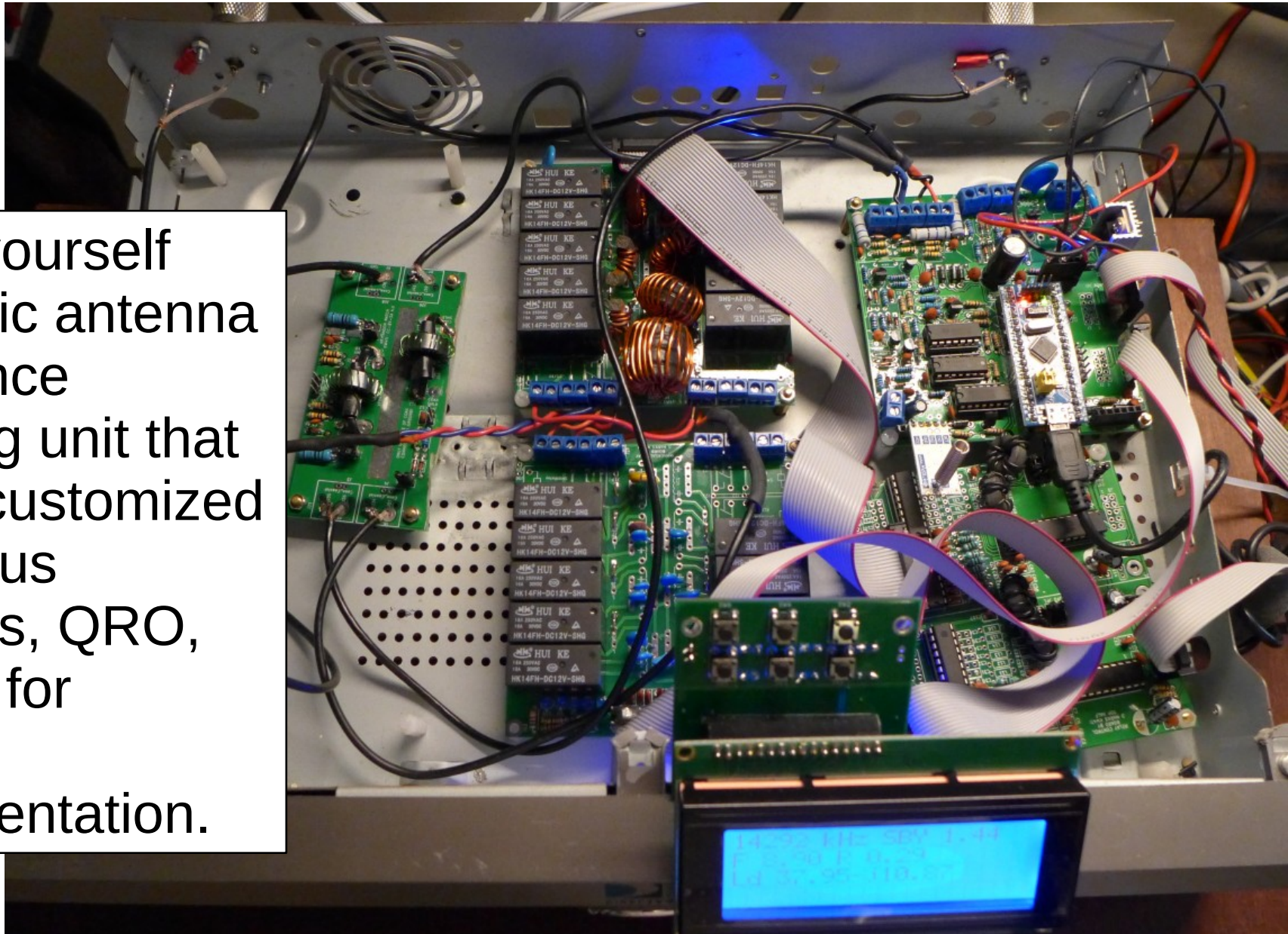
Examples from my open source projects: a Theremin

A microcontroller based Theremin, which is a musical instrument that can be played by moving one's hands near the instrument's antennas. Makes spooky sounds like in sci-fi movies.



Examples from my open source projects: an automatic antenna impedance matching unit

A do-it-yourself automatic antenna impedance matching unit that can be customized for various purposes, QRO, QRP, or for antenna experimentation.



New PCB manufacturing services make obtaining custom PCBs for your project very cheap!

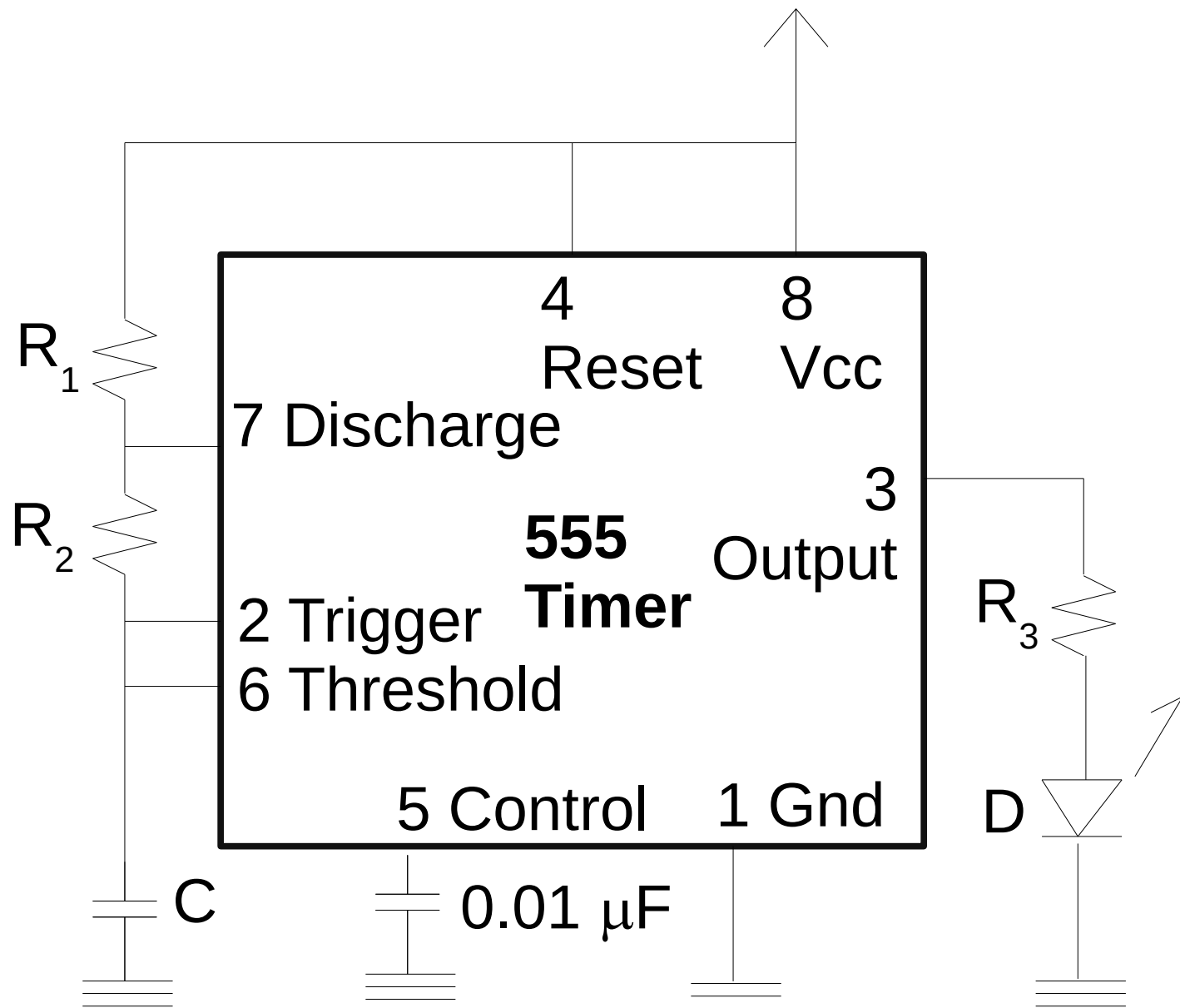
PCB manufacturers such as JLCPCB, Seeedstudio, and PCBway can manufacture two-layer PCBs up to 10-by-10 cm for \$5 or even less!

This can take the chore of handwiring together electronic components on a breadboard, and make it much easier and neater. They can also populate boards for you as well.

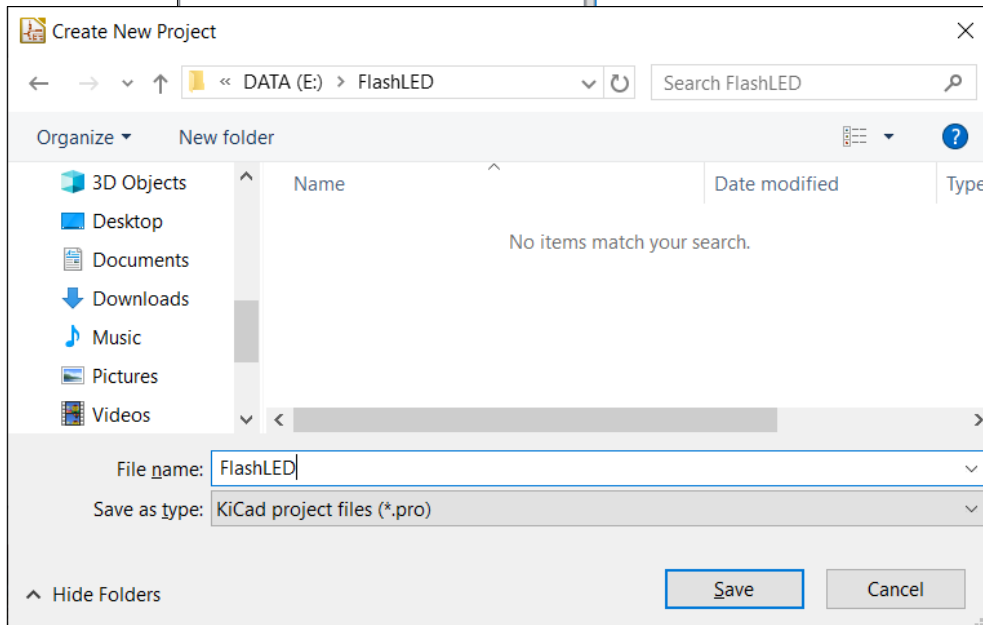
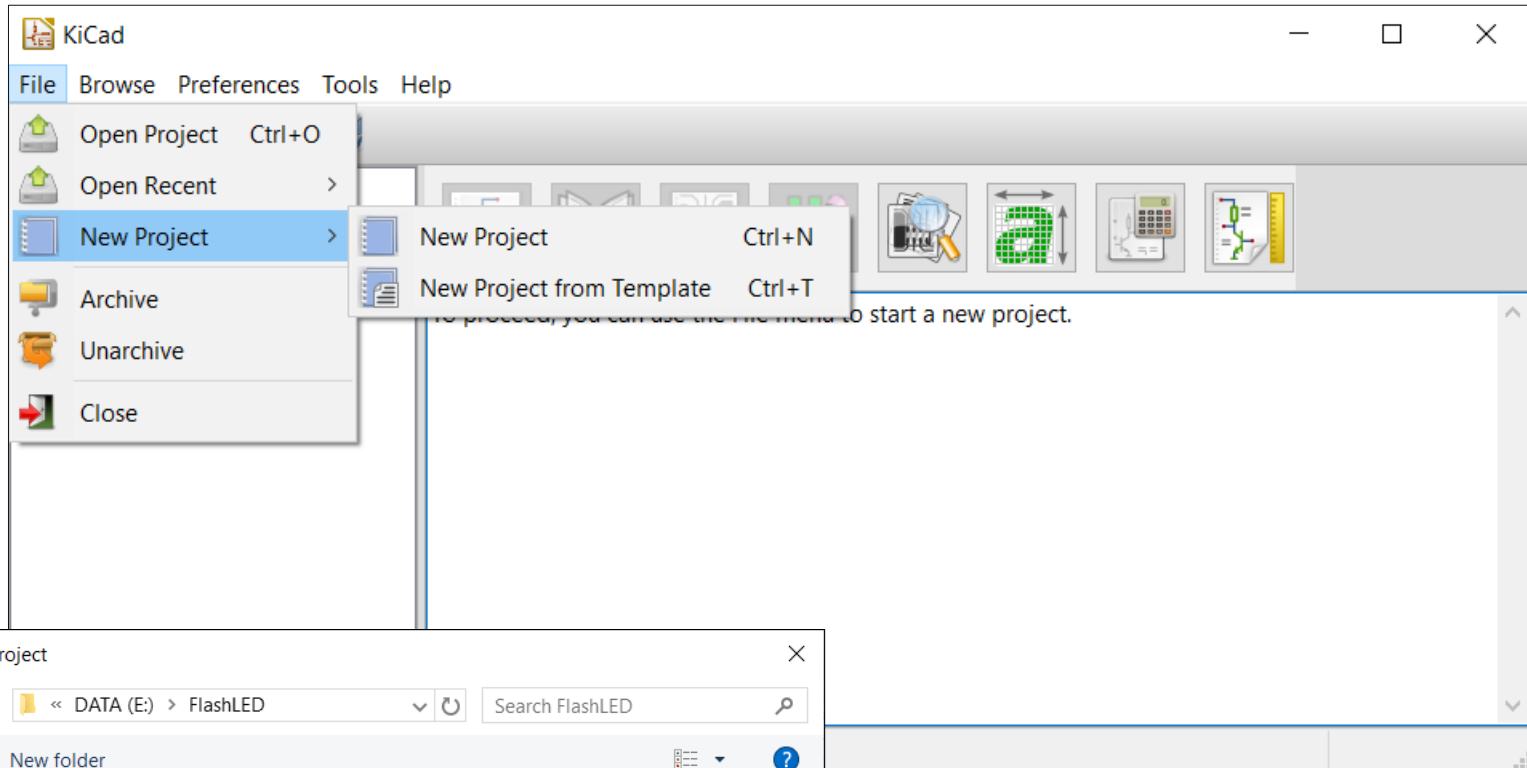


Today, I'm going to walk you through how to create a PCB with Kicad and submit it to one of these services.

Simple Kicad project: 555 LED flasher circuit



Starting a project



Create a directory for your project (for example “FlashLED”) and then place the project file “.pro” into that directory

The project is started. Now what?

File with your PCB layout

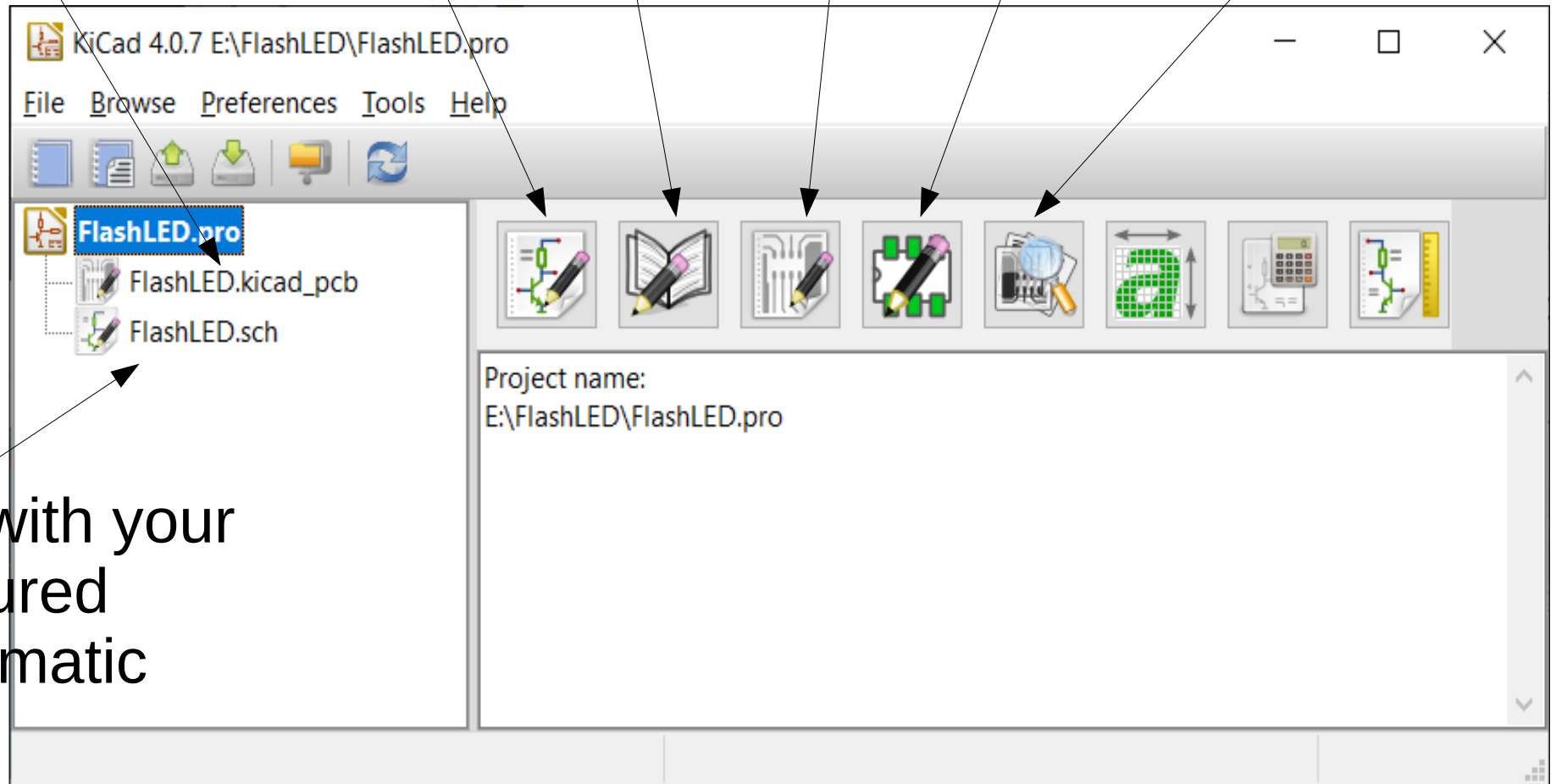
Schematic editor

Parts symbol editor

PCB Layout editor

Footprint editor

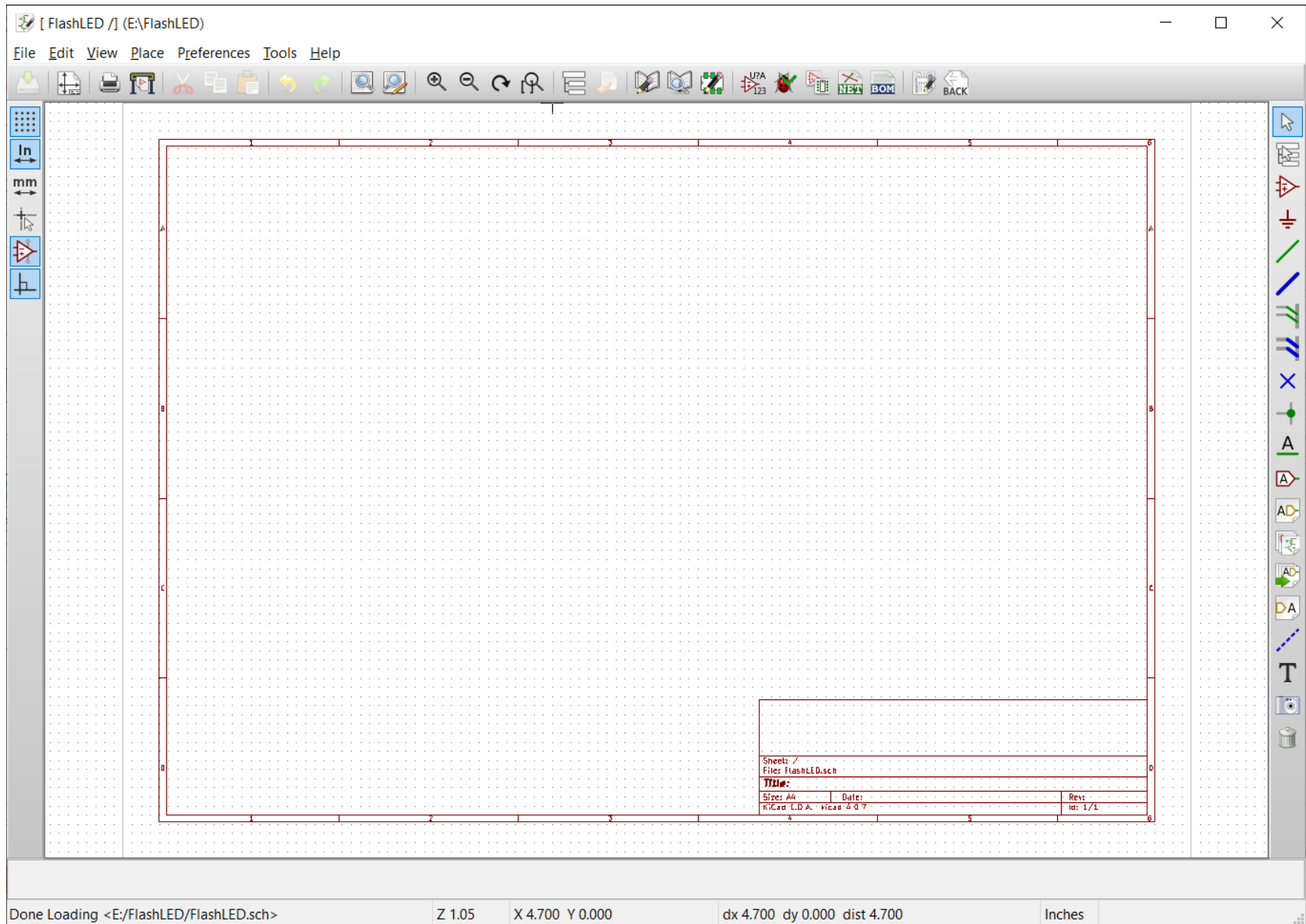
Gerber viewer



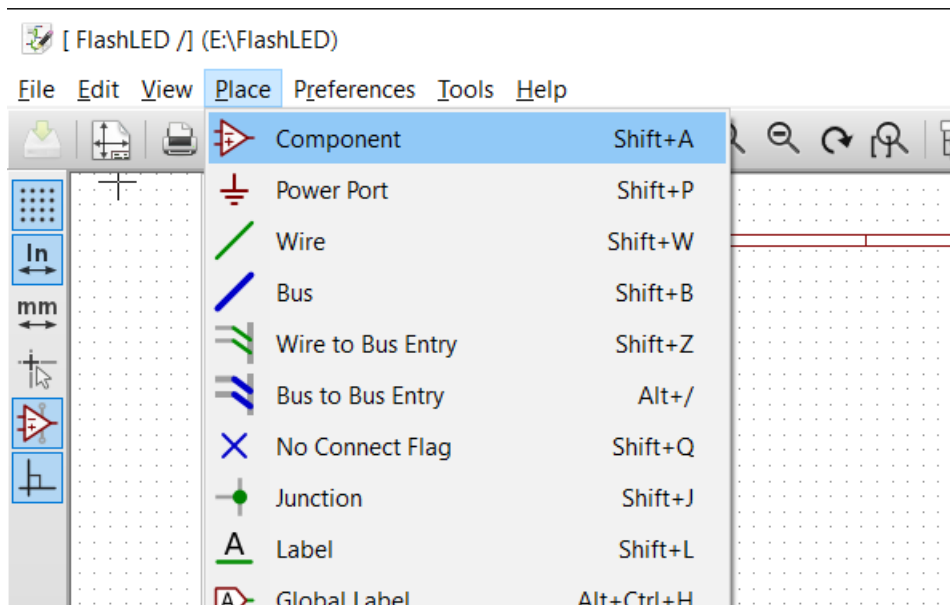
File with your captured schematic

First: capture schematic (schematic editor)

Empty schematic. Lets add some parts!

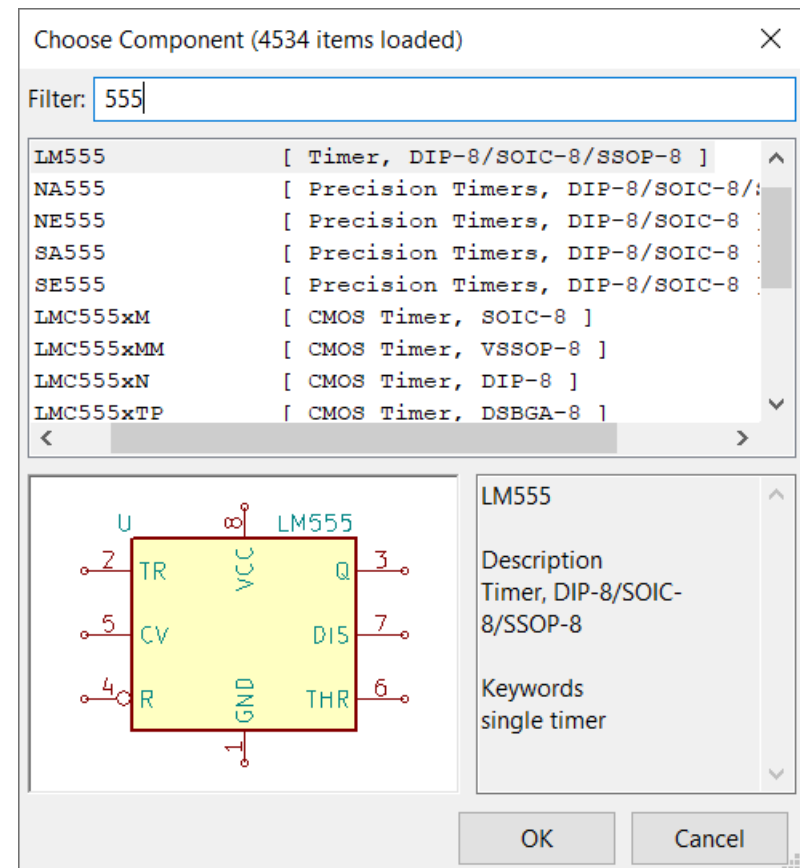


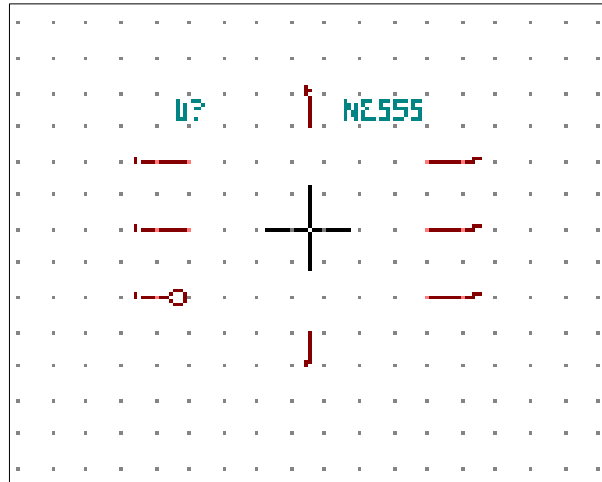
Place a component



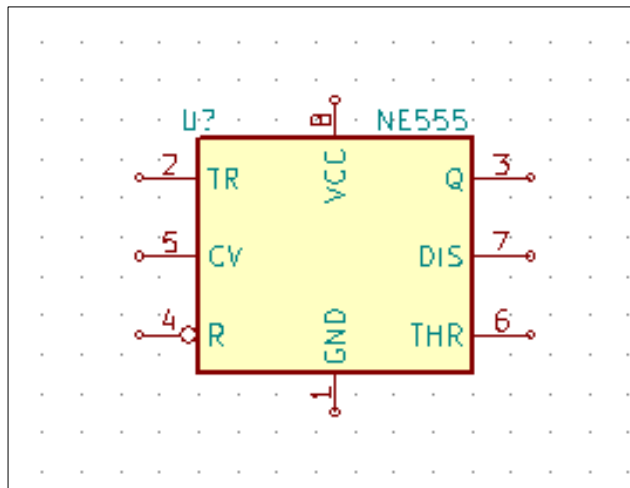
Then click on the location on the schematic you want the component placed with the crosshairs.

Type “555” to search the loaded libraries. Kicad lists the components with part numbers that match the search term.





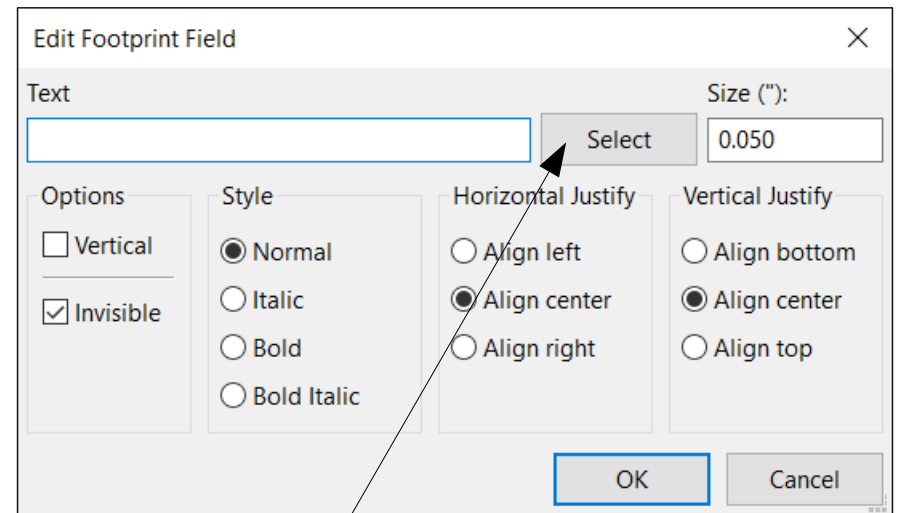
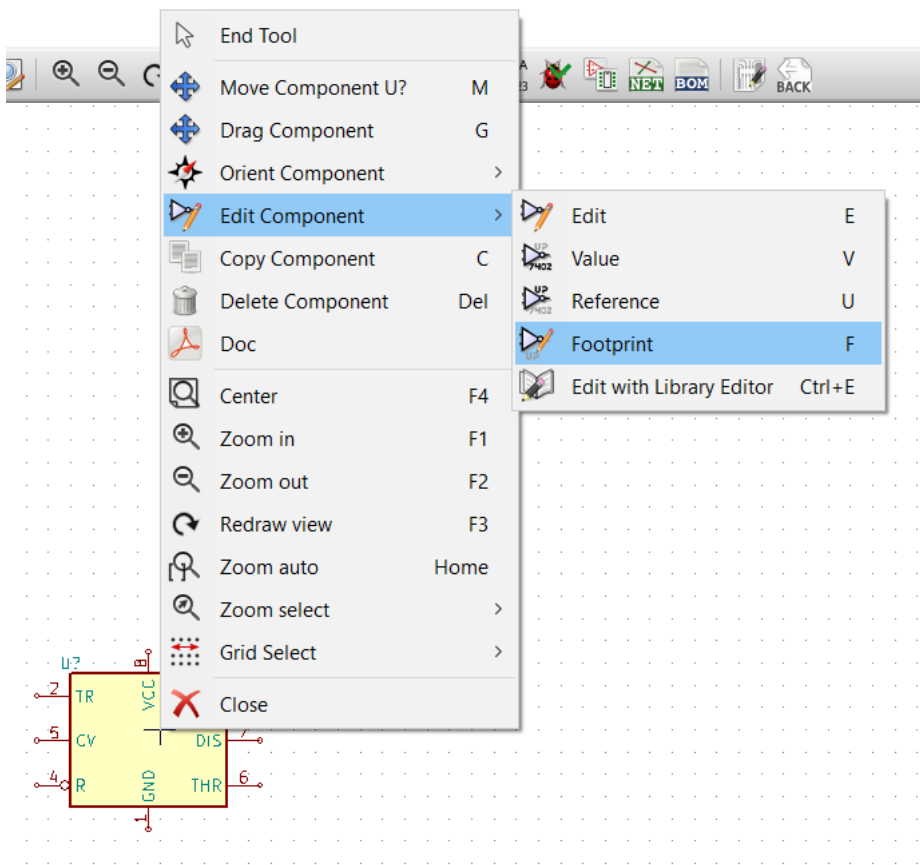
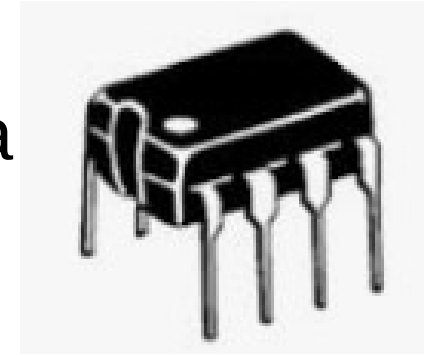
Move the part with the mouse cursor and click where the part should be placed.



Now the part is placed.

Assign a footprint

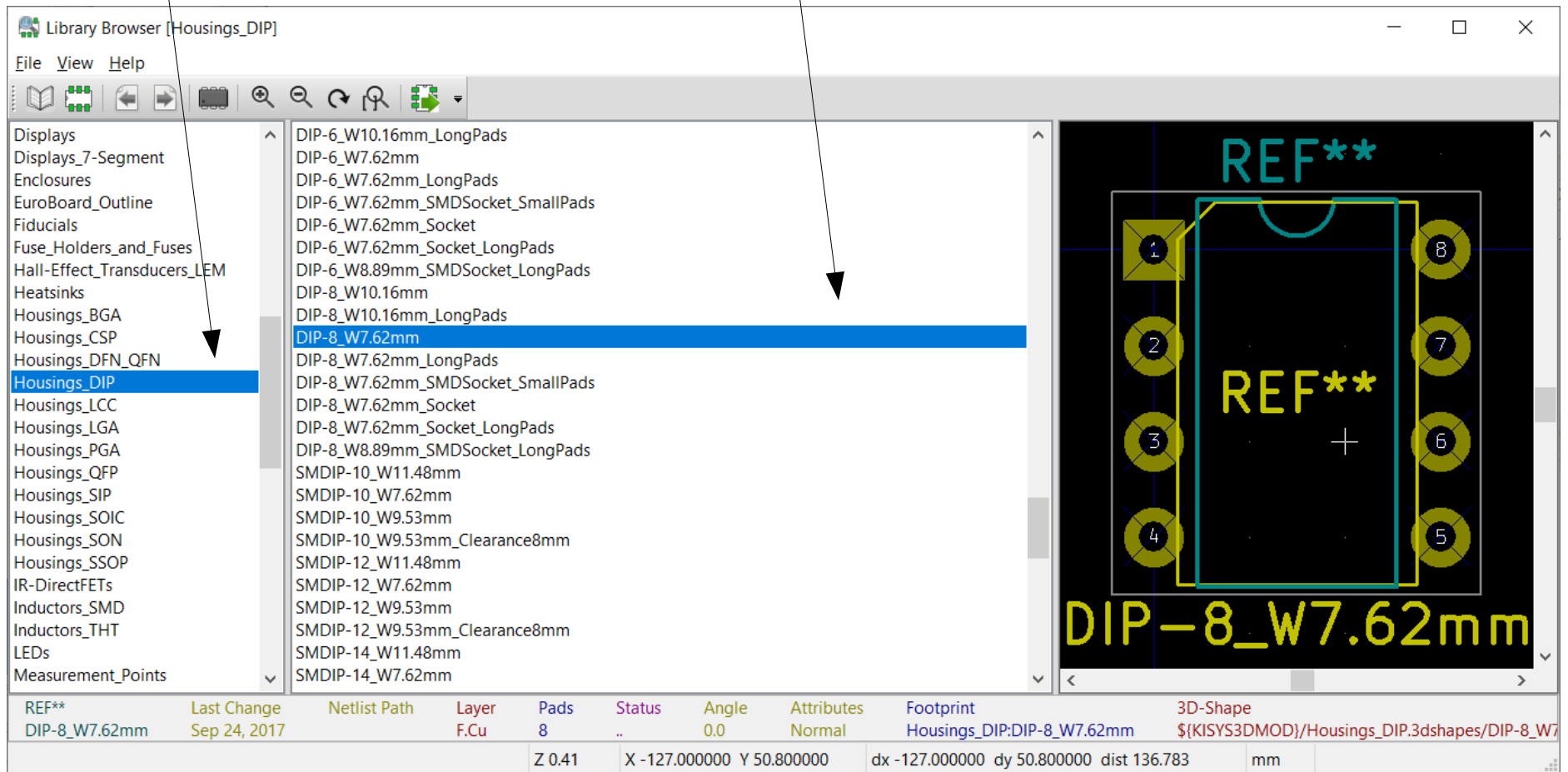
The footprint tells Kicad how to place the pads so that the selected part can be soldered into the board. For example, the NE555 chip has a DIP-8 package. To select the footprint, right-click on the part and then



Click Select to choose the footprint

Library with footprint

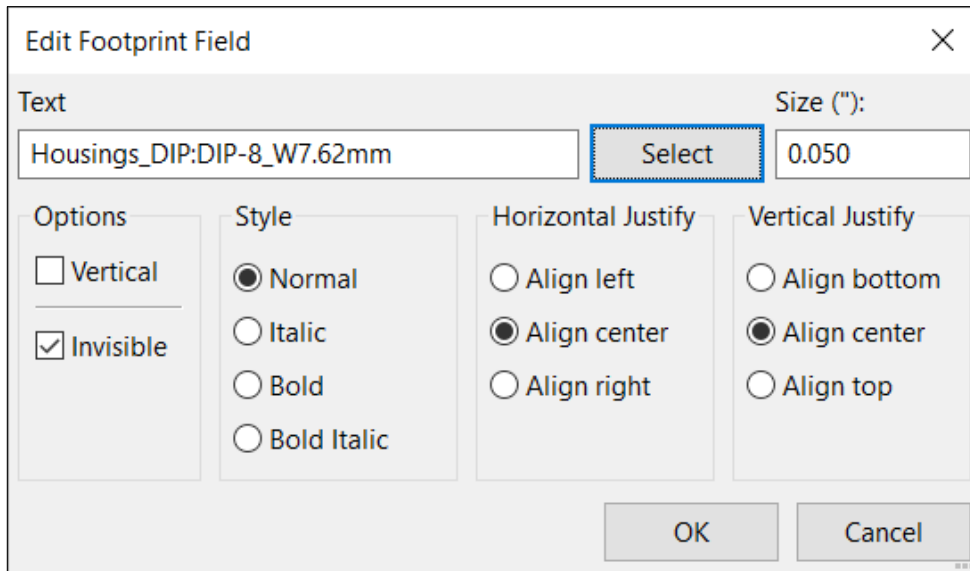
Selected footprint in library



Housings_DFN_QFN
Housings_DIP
Housings_LCC

DIP-8_W10.16mm_LongPads
DIP-8_W7.62mm
DIP-8_W7.62mm_LongPads

Footprint field now shows selected footprint



Click ok

Now you've placed your first part!

Placing a 1/4 watt resistor

RESISTORS_SMD

Resistors_THT

Resistors_Universal

R_Axial_DIN0204_L3.6mm_D1.6mm_P7.62mm_Horizontal

R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal

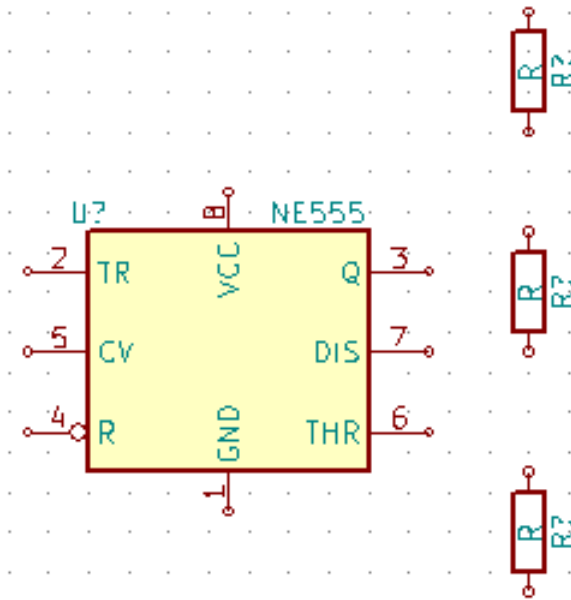
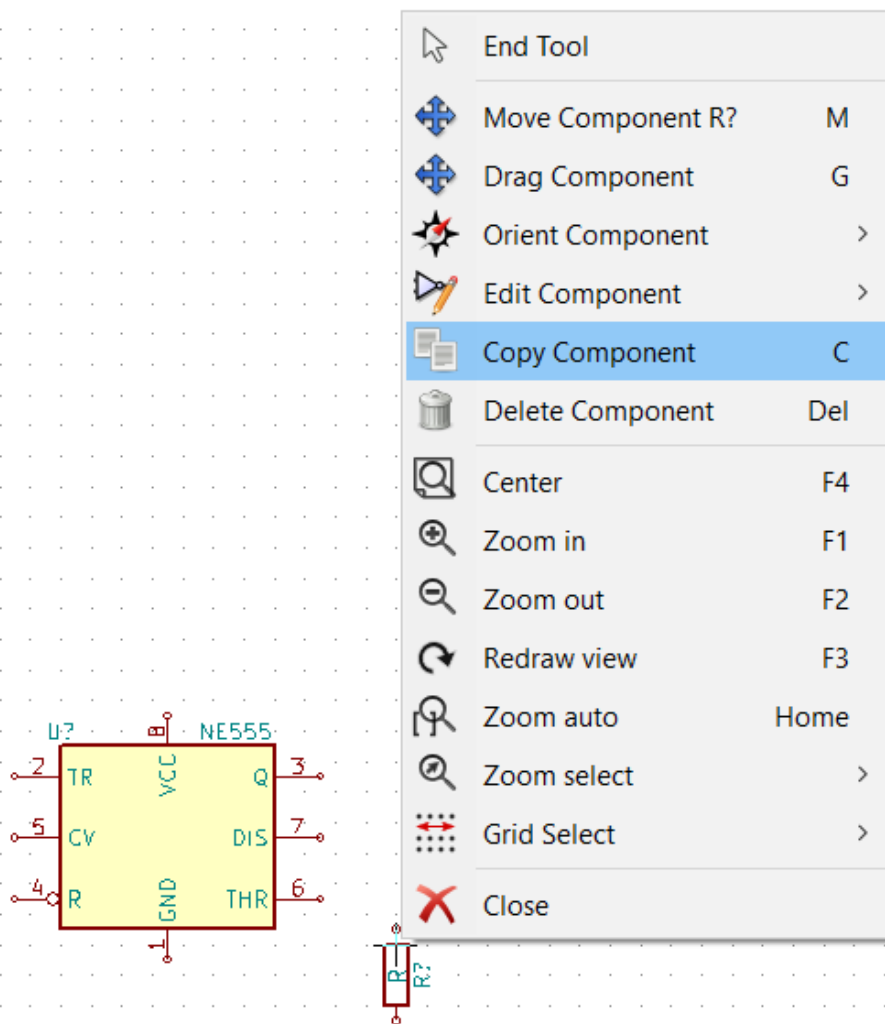
R_Axial_DIN0207_L6.3mm_D2.5mm_P15.24mm_Horizontal

The image shows a multi-step process in a PCB design software:

- Library Browser:** A tree view on the left shows the hierarchy: Resistor_THT > Resistor_Universal > Resistor_Axial > Resistor_Axial_DIN0207. The selected component is `R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal`.
- Choose Component Dialog:** A dialog box titled "Choose Component (4535 items loaded)" with a filter set to "R". It lists various resistor network components. The selected component is `R [Resistor]`. The dialog also shows a schematic symbol for a resistor and its description: "Resistor".
- Component Placement:** A window showing the selected resistor component being placed on a PCB grid. The component name `R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal` is visible in the placement area.
- Properties Table:** A table at the bottom of the software interface displays the properties of the placed component.

REF**	Last Change	Netlist Path	Layer	Pads	Status	Angle	Attributes	Footprint
R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal	Jan 10, 2017		F.Cu	2	..	0.0	Normal	Resistors_THT:R_Axial_DIN0207_L6.3mm_D2.5mm
		Z 9.82 X -2.540000 Y -2.540000			dx -2.540000 dy -2.540000 dist 3.592			mm

Now if we want two more resistors of the same type, we can just copy the one resistor twice



These all have the same $\frac{1}{4}$ watt package footprint we just selected.

Placing a small ceramic disc capacitor

Capacitors_SMD

Capacitors_THT

C_Disc_D5.0mm_W2.5mm_P2.50mm

C_Disc_D5.0mm_W2.5mm_P5.00mm

Library Browser [Capacitors_THT]

File View Help

- Buttons_Switches_THT
- Buzzers_Beepers
- Capacitors_SMD
- Capacitors_THT**
- Capacitors_Tantalum_SMD
- Connectors
- Connectors_Card
- Connectors_HDMI
- Connectors_Harwin
- Connectors_Hirose
- Connectors_JEC_DIN
- Connectors_JAE
- Connectors_JST
- Connectors_Mini-Universal
- Connectors_Molex
- Connectors_Multicomp
- Connectors_Phoenix
- Connectors_Samtec
- Connectors_TE-Connectivity
- Connectors_Terminal_Blocks
- Connectors_USB
- Connectors_WAGO
- Converters_DCDC_ACDC
- Crystals
- Diodes_SMD

C_Disc_D10.5mm_W5.0mm_P7.50mm
C_Disc_D11.0mm_W5.0mm_P10.00mm
C_Disc_D11.0mm_W5.0mm_P5.00mm
C_Disc_D11.0mm_W5.0mm_P7.50mm
C_Disc_D12.0mm_W4.4mm_P7.75mm
C_Disc_D12.5mm_W5.0mm_P10.00mm
C_Disc_D12.5mm_W5.0mm_P7.50mm
C_Disc_D14.5mm_W5.0mm_P10.00mm
C_Disc_D14.5mm_W5.0mm_P7.50mm
C_Disc_D16.0mm_W5.0mm_P10.00mm
C_Disc_D16.0mm_W5.0mm_P7.50mm
C_Disc_D3.0mm_W1.6mm_P2.50mm
C_Disc_D3.0mm_W2.0mm_P2.50mm
C_Disc_D3.4mm_W2.1mm_P2.50mm
C_Disc_D3.8mm_W2.6mm_P2.50mm
C_Disc_D4.3mm_W1.9mm_P5.00mm
C_Disc_D4.7mm_W2.5mm_P5.00mm
C_Disc_D5.0mm_W2.5mm_P2.50mm
C_Disc_D5.0mm_W2.5mm_P5.00mm
C_Disc_D5.1mm_W3.2mm_P5.00mm
C_Disc_D6.0mm_W2.5mm_P5.00mm
C_Disc_D6.0mm_W4.4mm_P5.00mm
C_Disc_D7.0mm_W2.5mm_P5.00mm
C_Disc_D7.5mm_W2.5mm_P5.00mm
C_Disc_D7.5mm_W4.4mm_P5.00mm

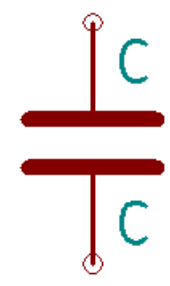
REF**	Last Change	Netlist Path	Layer	Pads	Status	Angle	Attributes	Footprint	3D-Shape
C_Disc_D5.0mm_W2.5mm_P5.00mm	Jul 28, 2017		F.Cu	2	..	0.0	Normal	Capacitors_THT:C_Disc_D5.0mm_W2.5mm_P5.00mm	\$(KISYS3DM

Z 2.87 X -15.240000 Y -10.160000 dx -15.240000 dy -10.160000 dist 18.316 mm

Choose Component (4536 items loaded)

Filter: C

C	[Unpolarized capacitor]
C_Feedthrough	[feedthrough capacitor]
C_Small	[Unpolarized capacitor]
C_Variable	[Variable capacitor]
CP	[Polarised capacitor]
CP1	[Polarised capacitor]
CP1_Small	[Polarised capacitor]
CP_Small	[Polarised capacitor]
Crystal	[Two pin crystal]



C

Description
Unpolarized capacitor

Keywords
cap capacitor

OK Cancel

REF**

REF**

C_Disc_D5.0mm_W2.5mm_P5.00mm

Placing a radial polarized capacitor

Capacitors_SMD

Capacitors_THT

Capacitors_Tantalu

CP_Radial_D40.0mm_P10.00mm

CP_Radial_D5.0mm_P2.00mm

CP_Radial_D5.0mm_P2.50mm

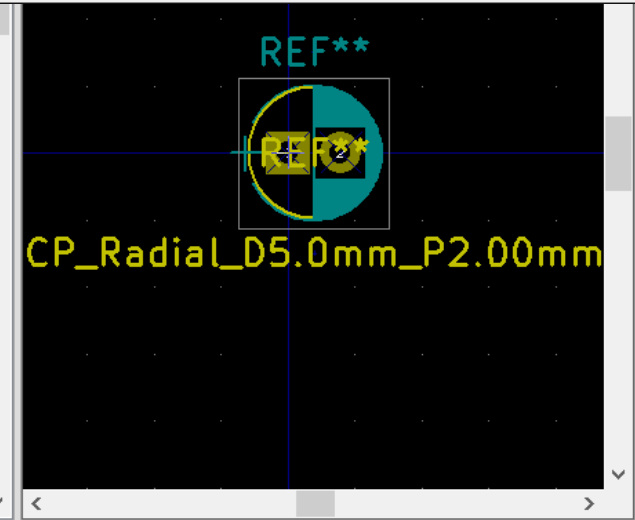
The screenshot shows the software interface for placing a capacitor. The Library Browser on the left shows the 'Capacitors_THT' category selected. The component list on the right includes 'CP_Radial_D5.0mm_P2.00mm', which is highlighted. The 'Choose Component' dialog box is open, showing the selected component 'CP' (Polarised capacitor) and its footprint 'CP_Radial_D5.0mm_P2.00mm'. The 3D model at the bottom right shows the capacitor footprint on a PCB, with a 'REF**' label and a 'CP_Radial_D5.0mm_P2.00mm' label.

REF**	Last Change	Netlist Path	Layer	Pads	Status	Angle	Attributes	Footprint	3D-Shape
CP_Radial_D5.0mm_P2.00mm	Jul 28, 2017		F.Cu	2	..	0.0	Normal	Capacitors_THT:CP_Radial_D5.0mm_P2.00mm	\$(KISYS3DMOD)/Capacit

The 'Choose Component' dialog box shows a list of components. The 'CP' component is selected, which is a 'Polarised capacitor'. The description and keywords for this component are also visible.

Component	Description
C_Small	[Unpolarized capacitor]
C_Variable	[Variable capacitor]
CP	[Polarised capacitor]
CP1	[Polarised capacitor]
CP1_Small	[Polarised capacitor]
CP_Small	[Polarised capacitor]
Crystal	[Two pin crystal]
Crystal_GND2	[Three pin crystal (GND on p
Crvstal GND23	[Four pin crystal (GND on pi

CP
Description
Polarised capacitor
Keywords
cap capacitor



Placing a 5 mm LED

INDUCTORS_THT
LEDs

LED_D5.0mm
LED_D5.0mm-3
LED_D5.0mm-4

Library Browser [LEDs]

File View Help

Housings_CSP
Housings_DFN_QFN
Housings_DIP
Housings_LCC
Housings_LGA
Housings_PGA
Housings_QFP
Housings_SIP
Housings_SOIC
Housings_SON
Housings_SSOP
IR-DirectFETs
Inductors_SMD
Inductors_THT
LEDs
Measurement_Points
Measurement_Scales
Microwave
Modules
Mounting_Holes
Opto-Devices
Oscillators
PFF_PSF_PSS_Leadforms
Pin_Headers
Potentiometers

LED_D3.0mm_Horizontal_O3.81mm_Z6.0mm
LED_D3.0mm_Horizontal_O6.35mm_Z10.0mm
LED_D3.0mm_Horizontal_O6.35mm_Z2.0mm
LED_D3.0mm_Horizontal_O6.35mm_Z6.0mm
LED_D4.0mm
LED_D5.0mm
LED_D5.0mm-3
LED_D5.0mm-4
LED_D5.0mm_FlatTop
LED_D5.0mm_Horizontal_O1.27mm_Z15.0mm
LED_D5.0mm_Horizontal_O1.27mm_Z3.0mm
LED_D5.0mm_Horizontal_O1.27mm_Z9.0mm
LED_D5.0mm_Horizontal_O3.81mm_Z15.0mm
LED_D5.0mm_Horizontal_O3.81mm_Z3.0mm
LED_D5.0mm_Horizontal_O3.81mm_Z9.0mm
LED_D5.0mm_Horizontal_O6.35mm_Z15.0mm
LED_D5.0mm_Horizontal_O6.35mm_Z3.0mm
LED_D5.0mm_Horizontal_O6.35mm_Z9.0mm
LED_D8.0mm
LED_D8.0mm-3
LED_Normandledd_WS2813-06_5.0x5.0mm_Pitch1.6mm
LED_Oval_W5.2mm_H3.8mm
LED_PLCC-2
LED_PLCC_2835
LED_PLCC_2835_Handsoldering

REF**	Last Change	Netlist Path	Layer	Pads	Status	Angle	Attributes	Footprint	3D-Shape	Doc: LED, c
LED_D5.0mm	Aug 17, 2017		F.Cu	2	..	0.0	Normal	LEDs:LED_D5.0mm	\$(KISYS3DMOD)/LEDs.3dshapes/LED_D5.0mm.wrl	Key Words

Z 5.60 X -10.160000 Y -2.540000 dx -10.160000 dy -2.540000 dist 10.473 mm

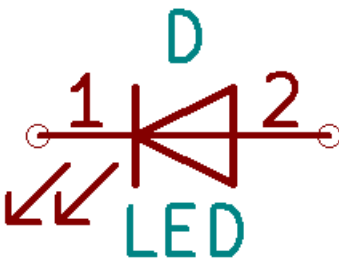
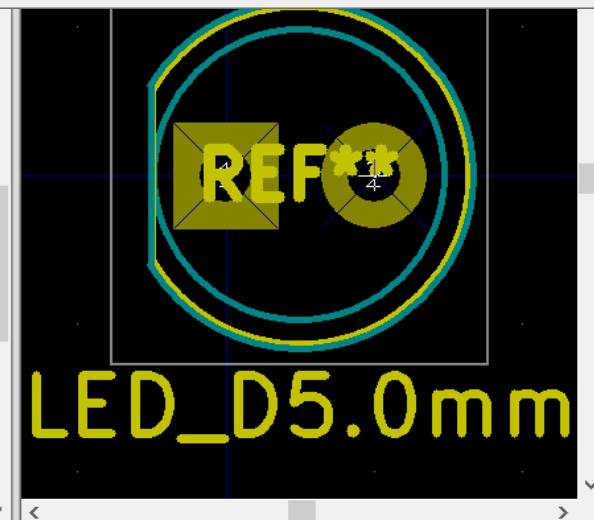
Choose Component (4538 items loaded)

Filter: LED

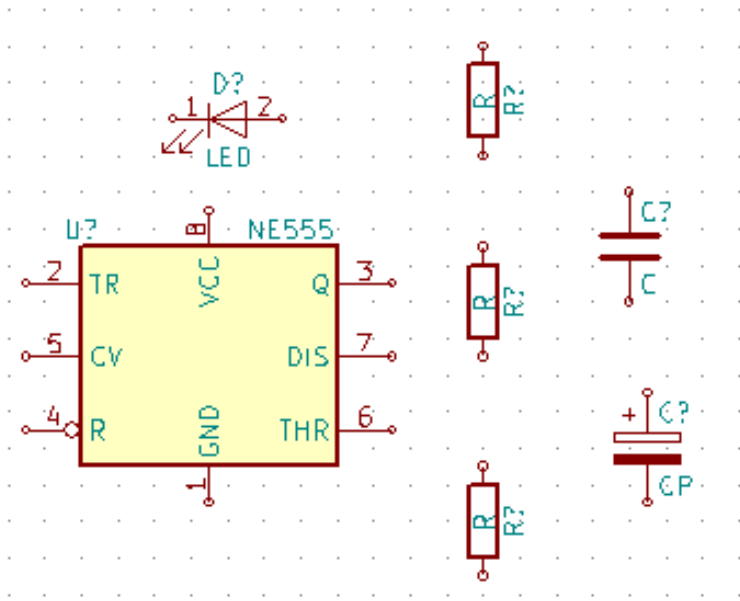
LED [LED generic]
LED_ALT [LED generic, alternativ symbol]
LED_ARGB [LED RGB, common anode (pin 1)]
LED_CRGB [LED RGB, Common Cathode]
LED_Dual_2pin [LED dual, 2pin version]
LED_Dual_AAC [LED dual, common cathode]
LED_Dual_AACC [LED dual, 4-pin]
LED_Dual_ACA [LED dual, common cathode]
LED Dual ACAC [LED dual, 4-pin]

LED
Description
LED generic
Keywords
led diode

OK Cancel

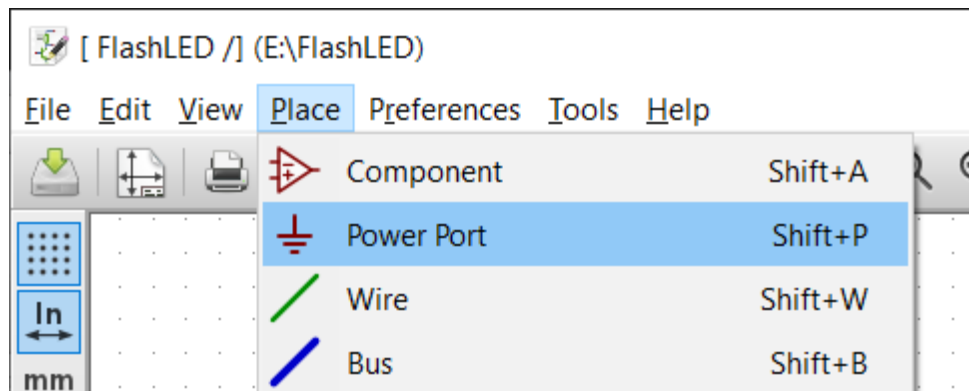
What we have so far...



What else do we need?

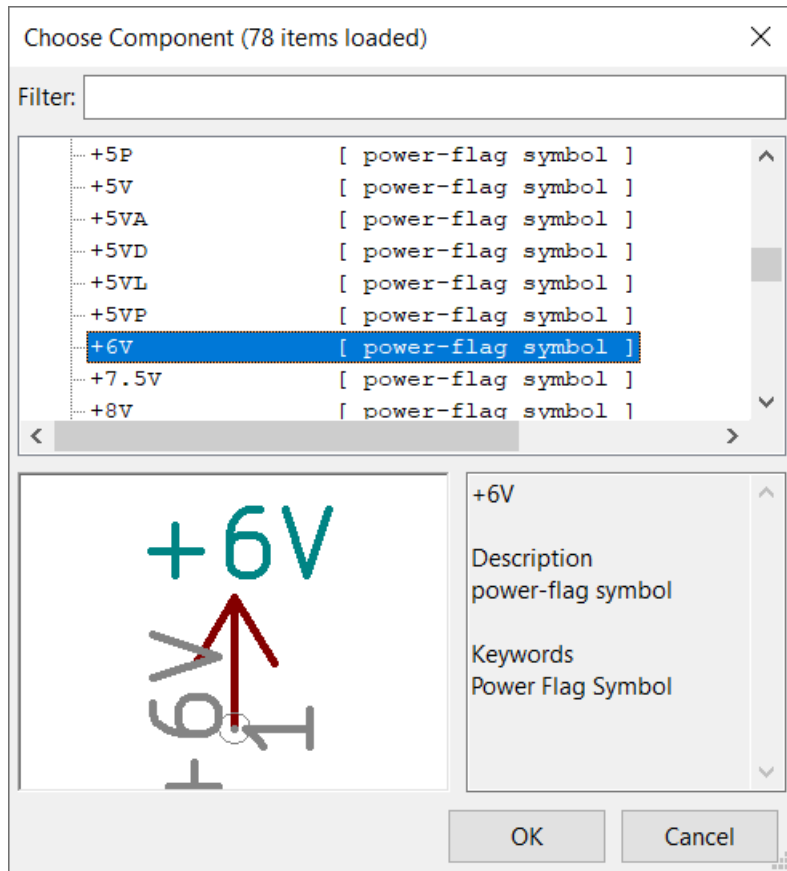
POWER!

To do this we need power ports.

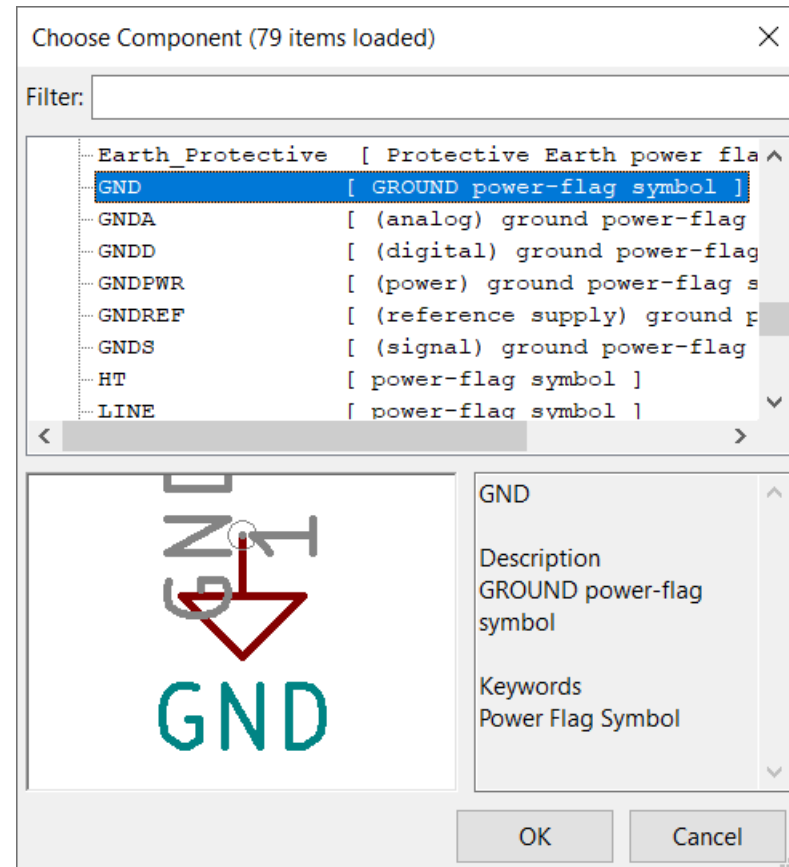


We choose the place power port menu item.

We can expand the list of power ports and select 6 volts, for example, for the positive voltage.

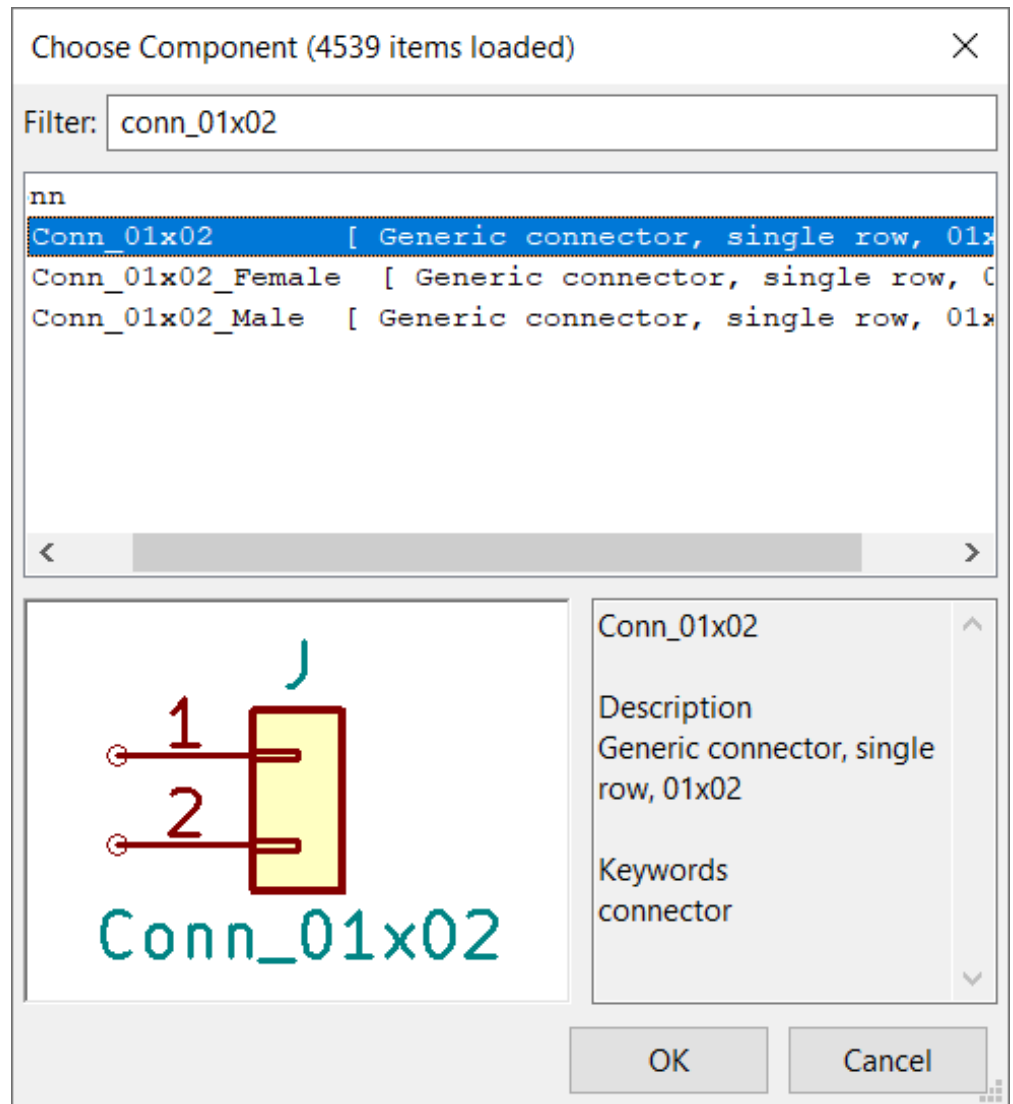
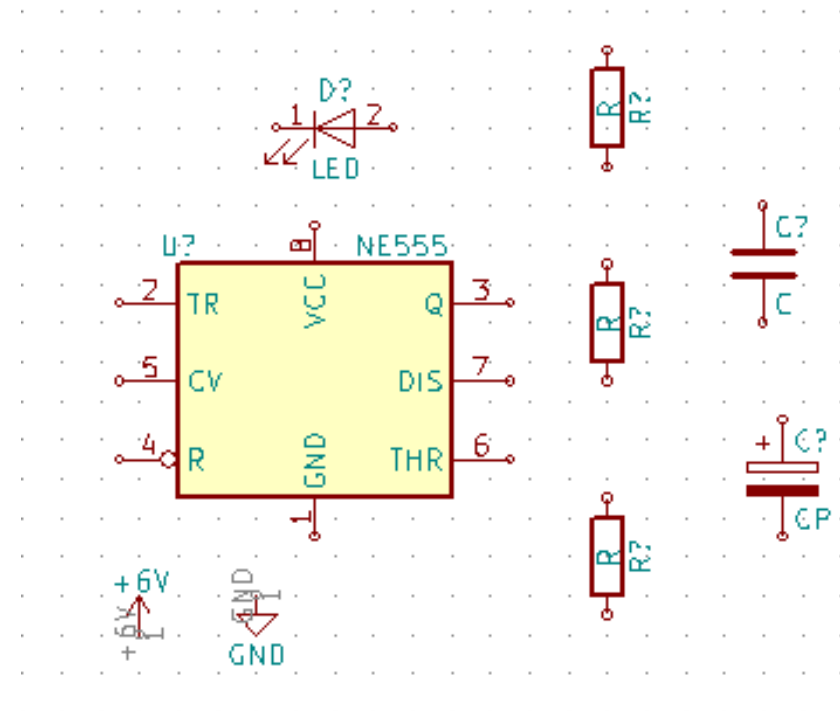


Then we can place a second power port for ground.



Now we have power ports. Finally, we need a way to connect external wires to the power port, so we place screw terminals.

Placing a two terminal connector

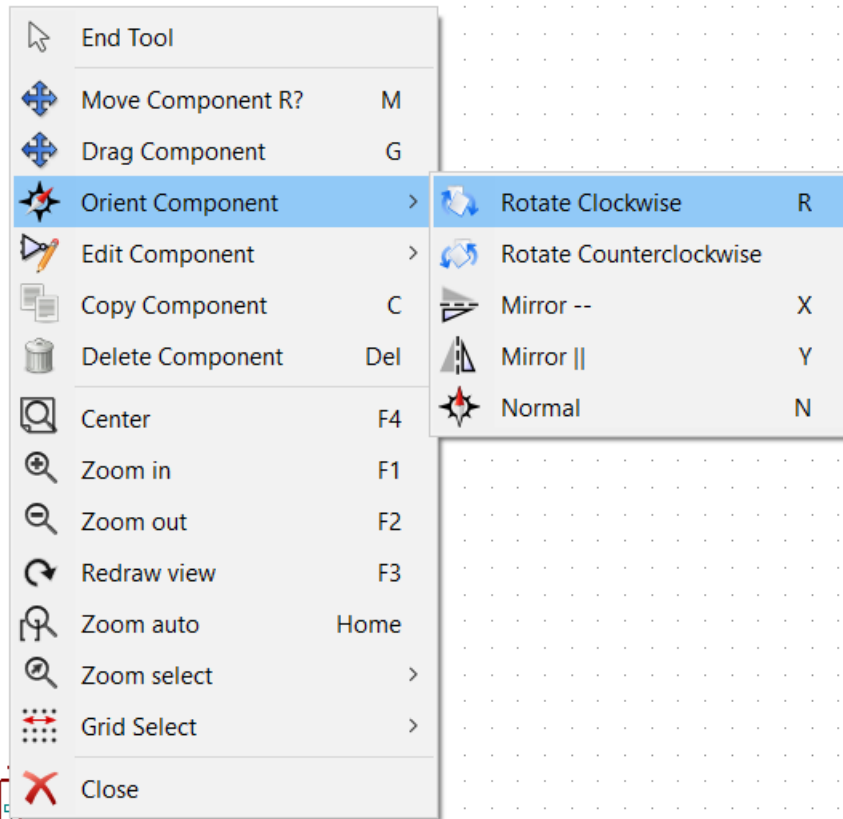
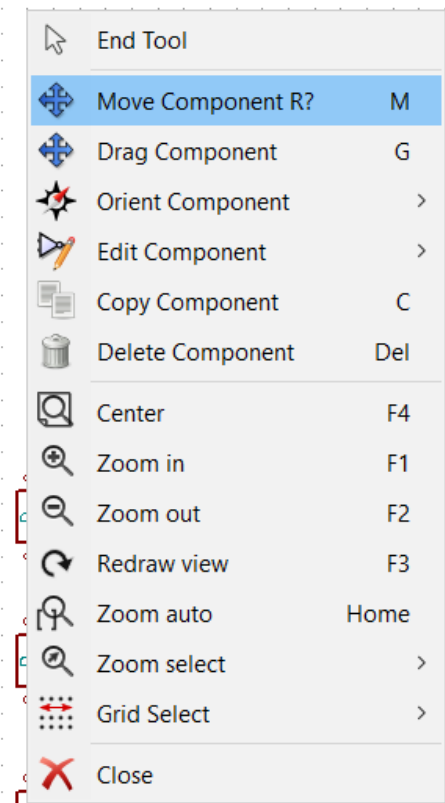
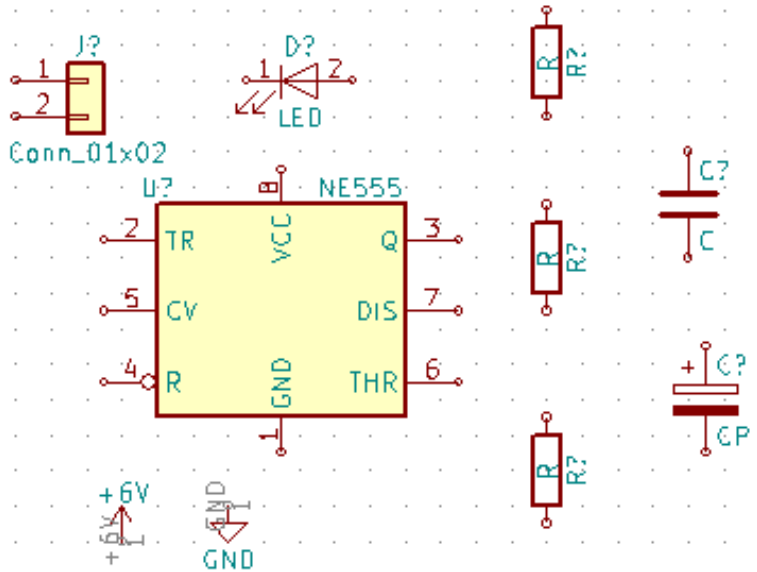


Connectors_TE-Connectivity
Connectors_Terminal_Blocks
Connectors_IISR

TerminalBlock_bornier-2_P5.08mm

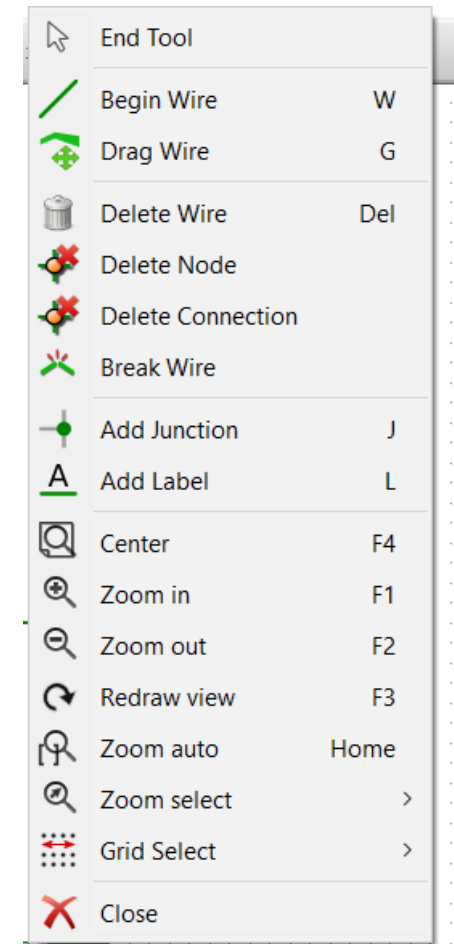
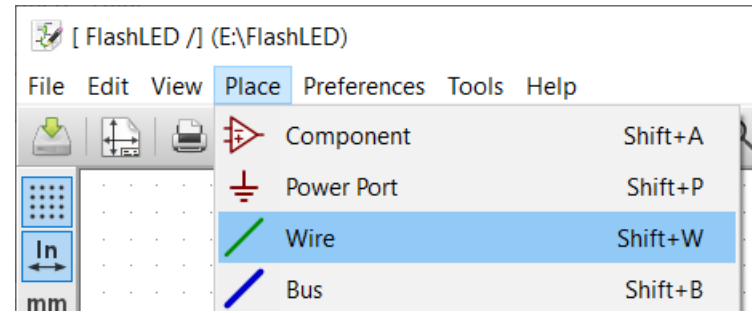
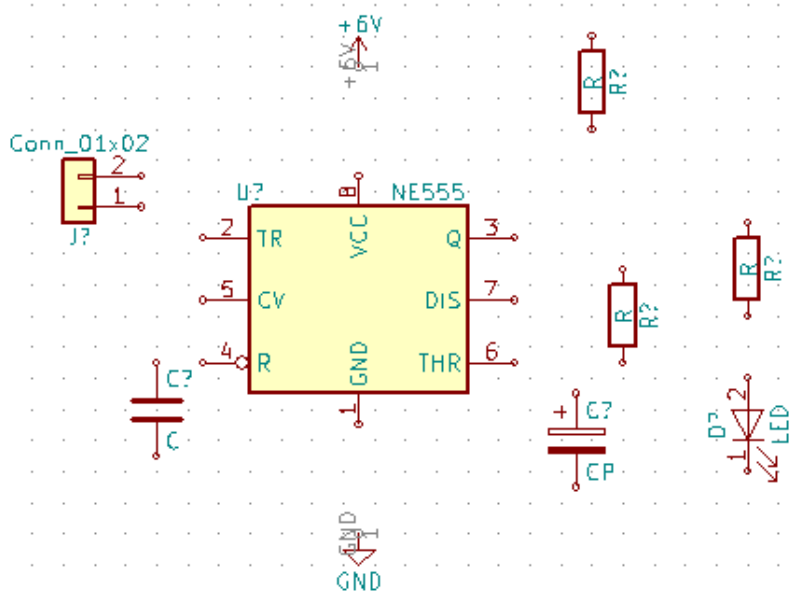
Footprint for standard 2/10" spacing screw terminals.

Wiring it all up



First move and rotate the components into the approximate right places. Right click on part and select move or rotate component.

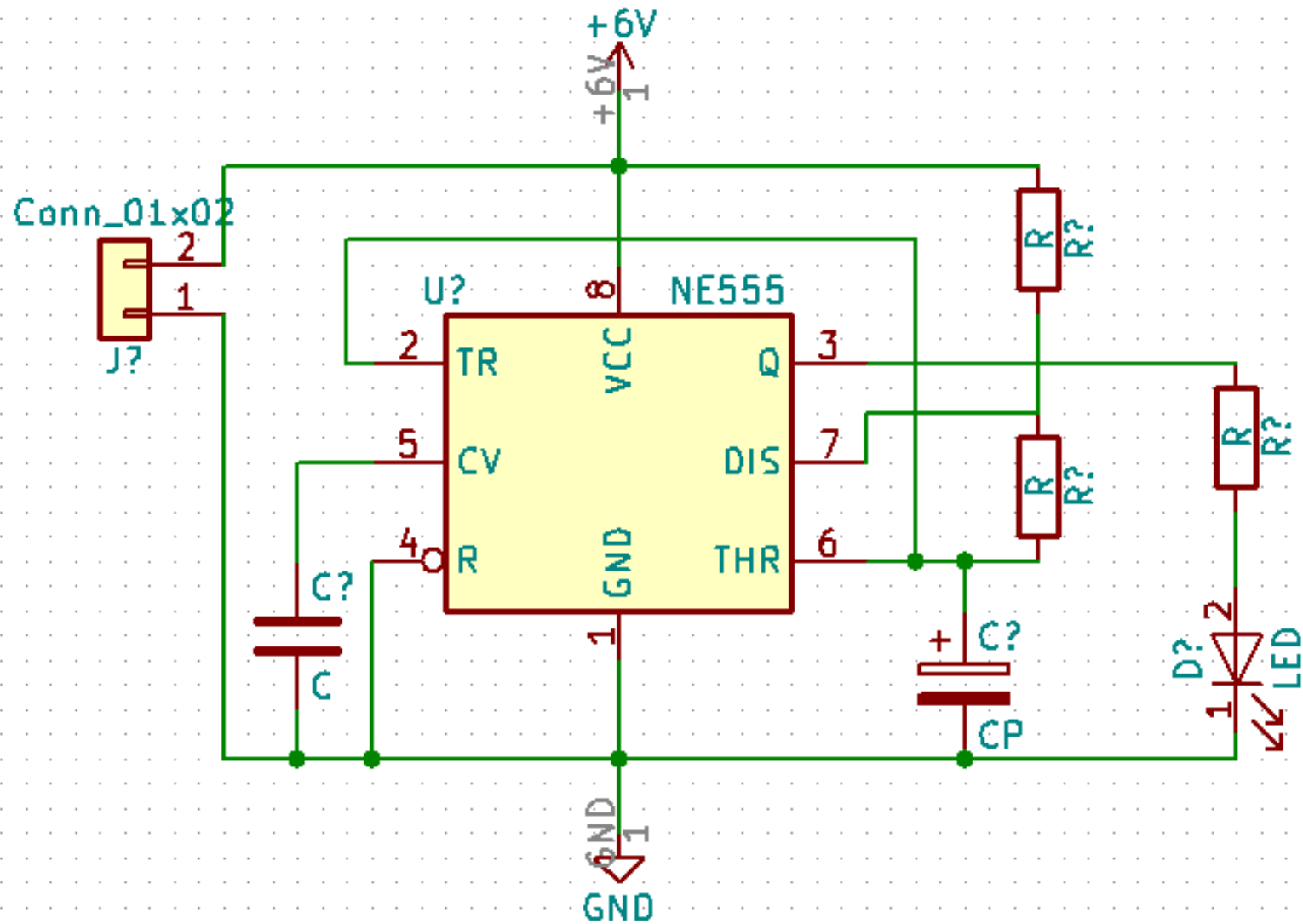
Now we can draw wires between the components



Click where you want wires to begin or end. You can place bends in wires by clicking on points you want the wire to bend at.

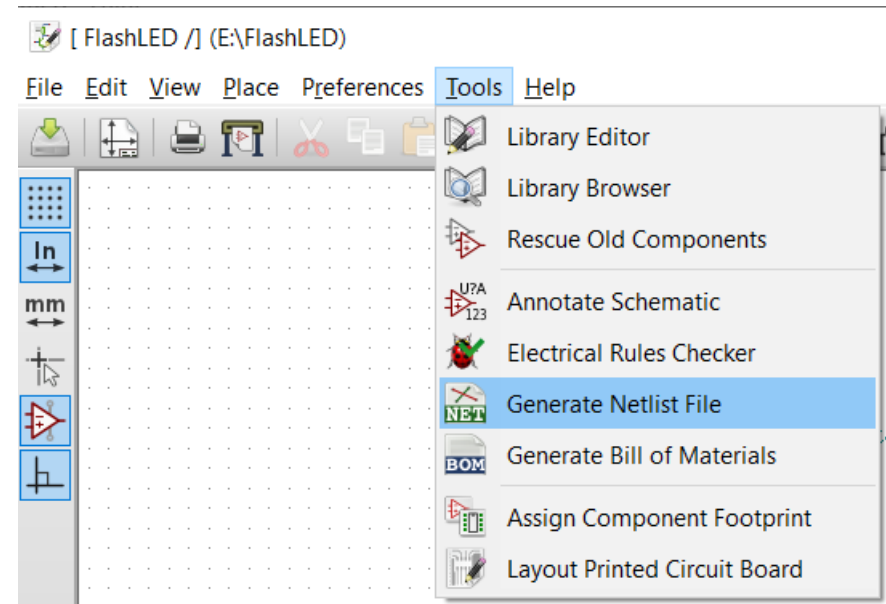
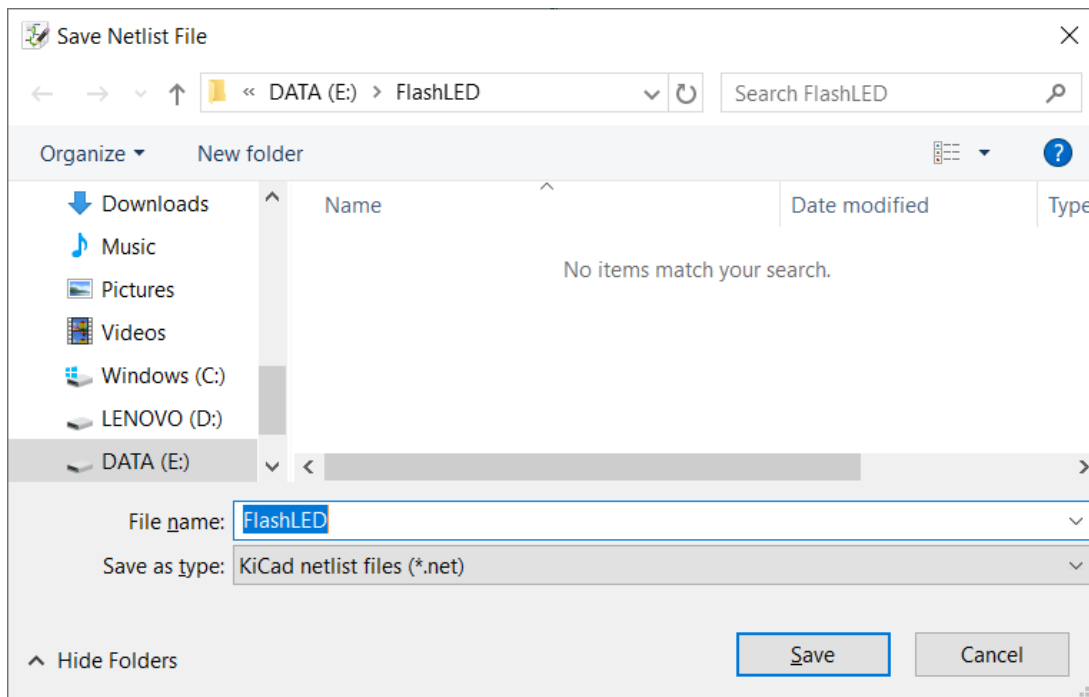
If you right-click on a wire, you can remove the connection.

Schematic capture is complete!



Schematic capture is complete!

Now we export a netlist, which provides the PCB layout editor the description of the schematic.



We select the name of the netlist file to save to (usually the default).

Annotating the schematic references

Since we didn't specify what the part references were (for example R1, R2, R3 for the three resistors), and these are required for the PCB editor, Kicad will automatically annotate the references. These may be assigned manually if desired as well when you create the part.

Annotate Schematic

Exporting the netlist requires a completely annotated schematic.

Scope

- Use the entire schematic
- Use the current page only


Keep existing annotation

Reset existing annotation

Reset, but do not swap any annotated multi-unit parts

Annotation Order

- Sort components by X position
- Sort components by Y position



Annotation Choice

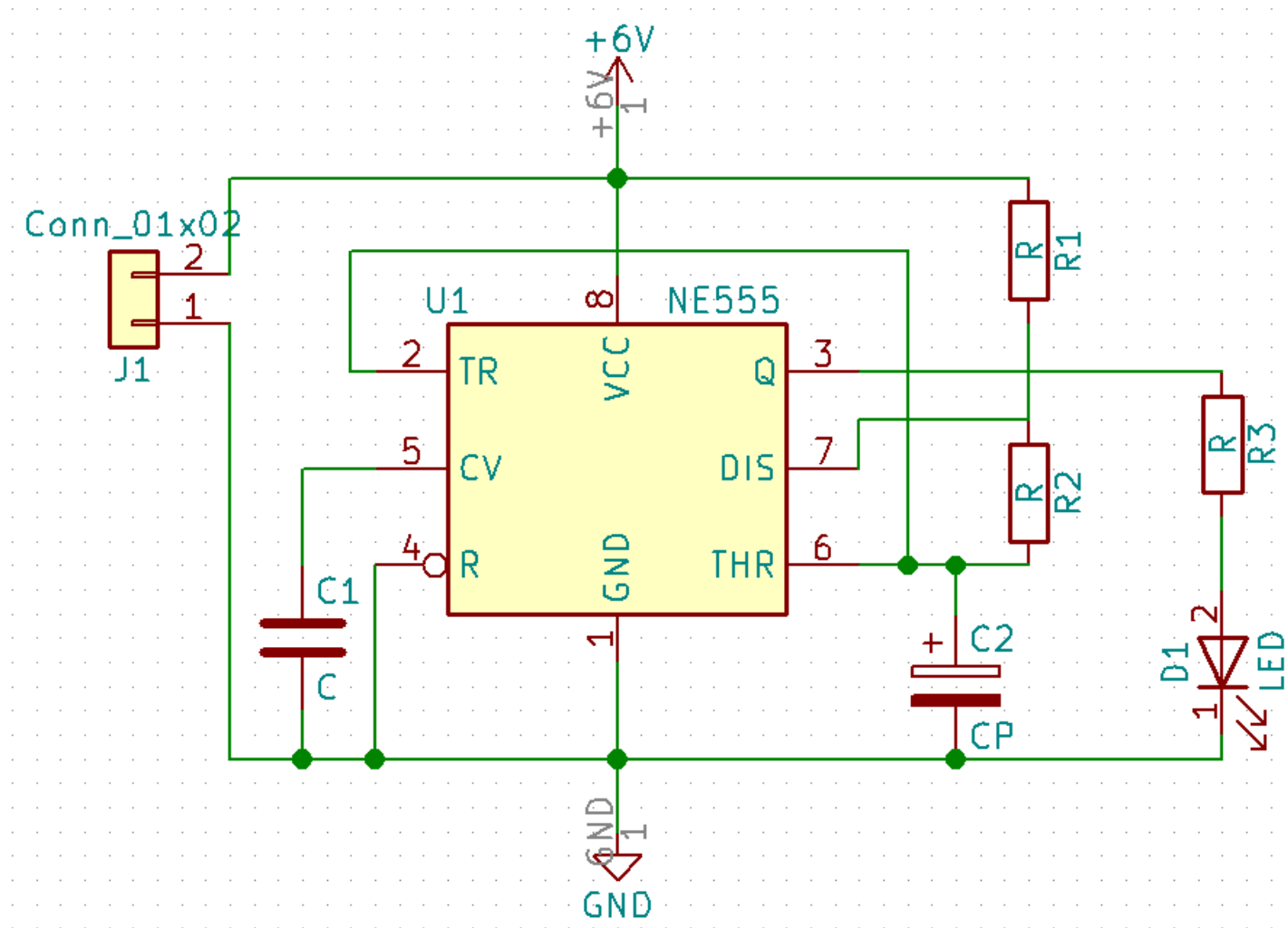
- Use first free number in schematic
- Start to sheet number*100 and use first free number
- Start to sheet number*1000 and use first free number

Dialog

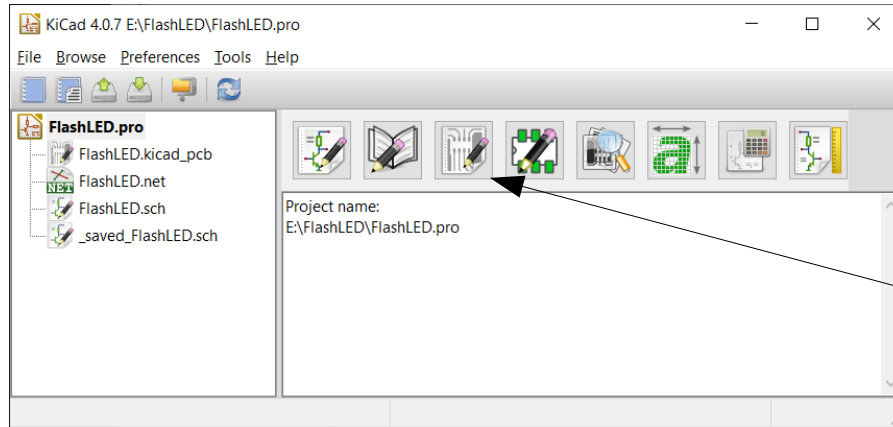
- Keep this dialog open
- Always ask for confirmation

Close Clear Annotation Annotate

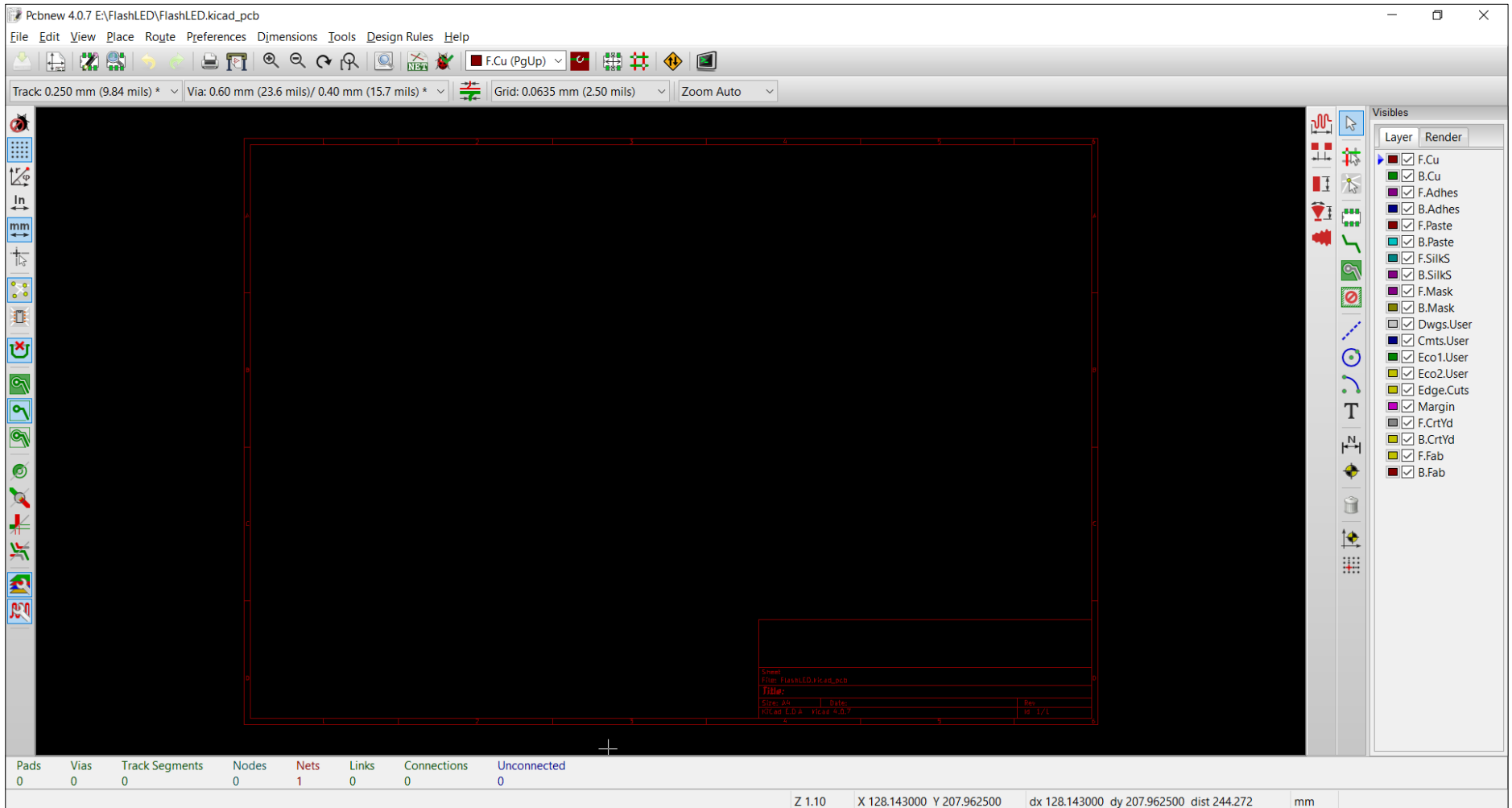
Now the references are added to the schematic.



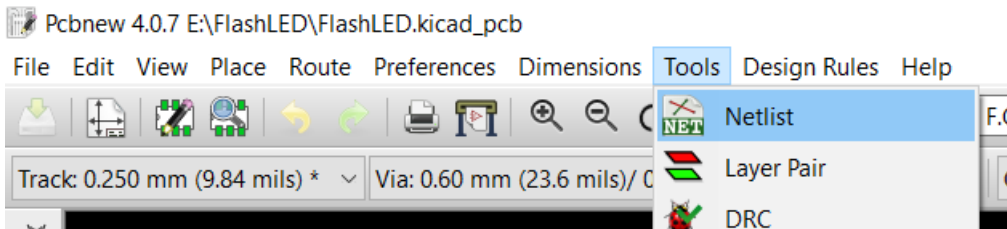
On with laying out the PCB!



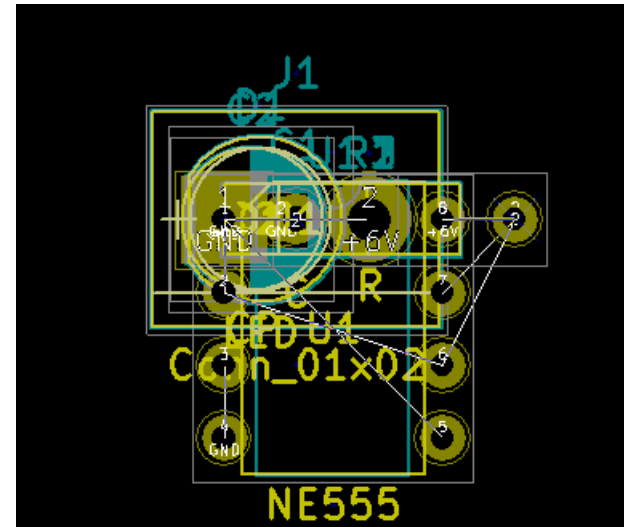
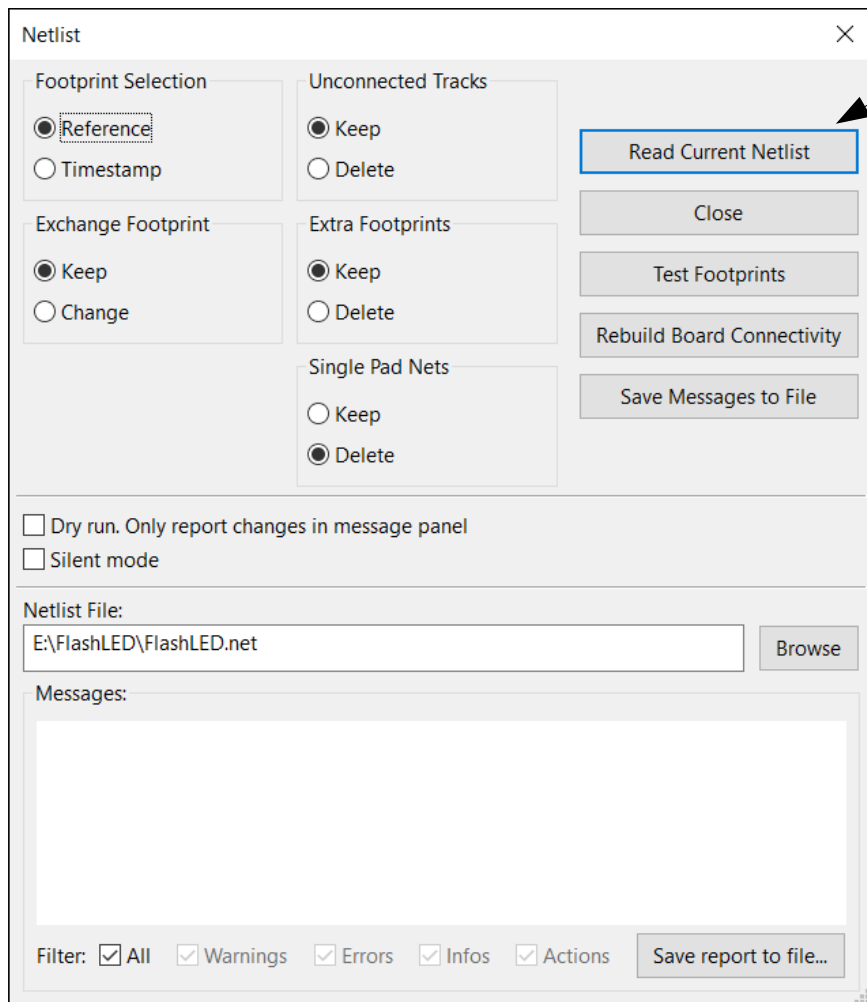
PCB
Layout
PCB Editor



Importing the netlist into the PCB editor



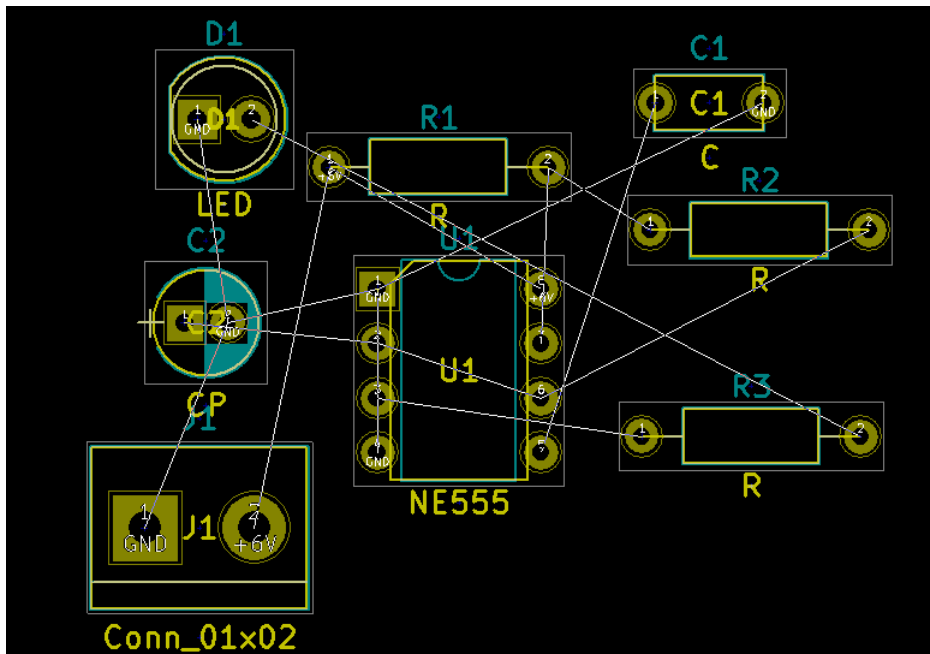
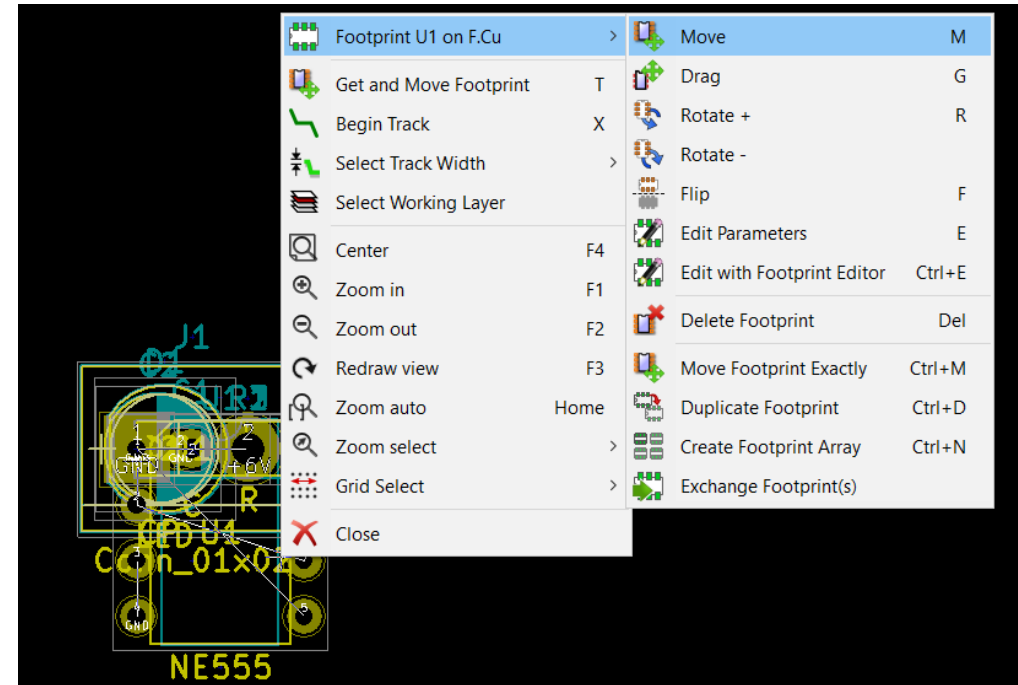
Click Read
Current Netlist



All of the components
are imported, and lines
drawn between the pins
to be connected.

Moving the components

Lets move the parts so they're not on top of each other so we can better see what we're doing.



Kicad shows us how the parts are to be connected, but these are not wires. This is called the “ratsnest.”

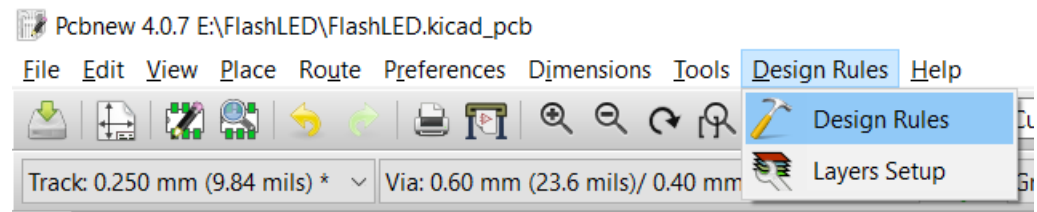
Design rules

We have to decide aspects such as

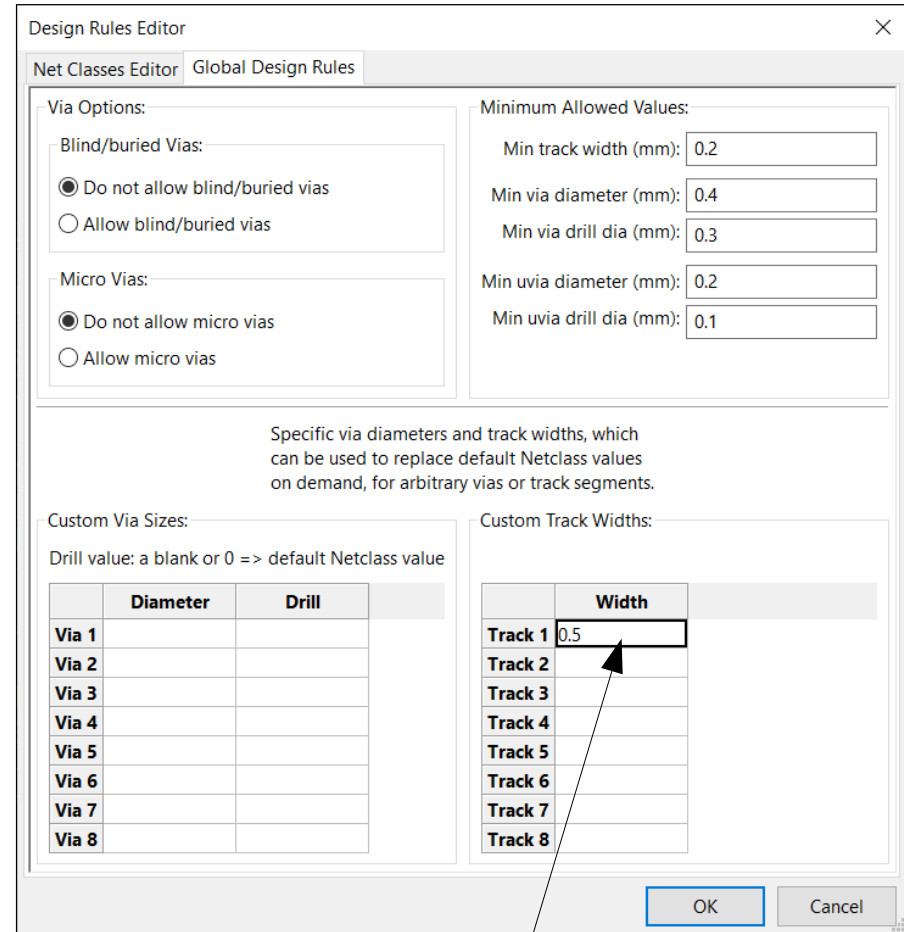
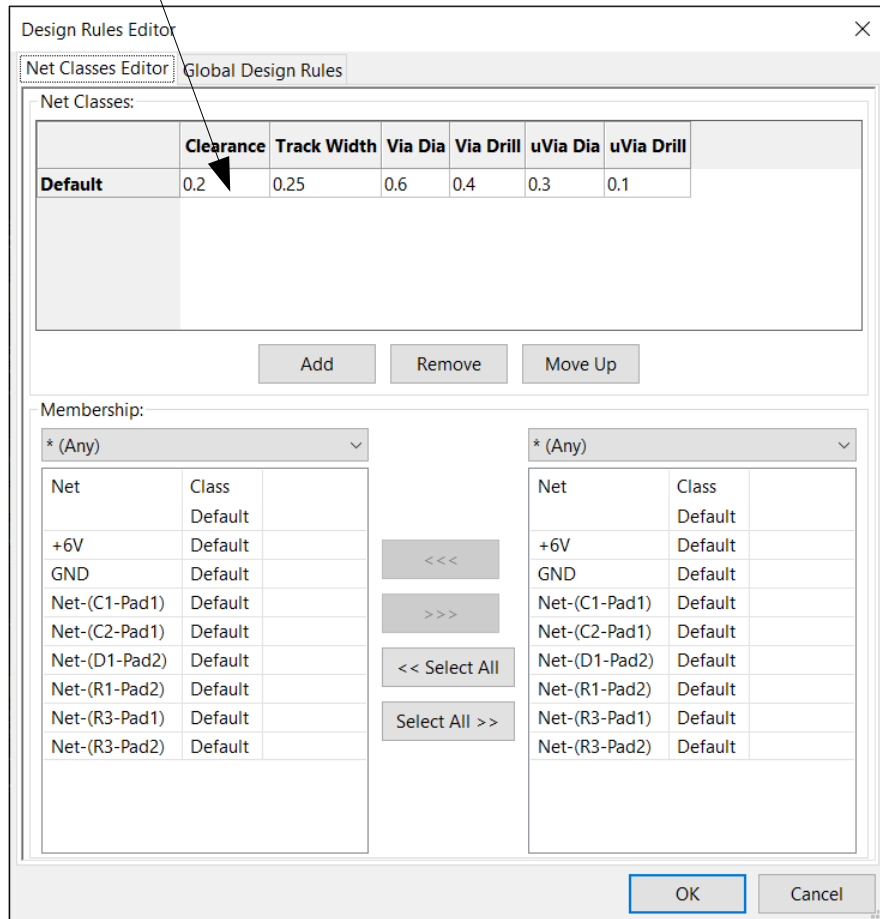
- How wide or narrow will the traces be between the components?
- How close are the traces allowed to be?
- How big or small are vias allowed to be?

These are often determined by the manufacturer of your PCB. If you want very small features, this costs more. Most hobbyist projects do not require very challenging or expensive design rules.

Setting the design rules

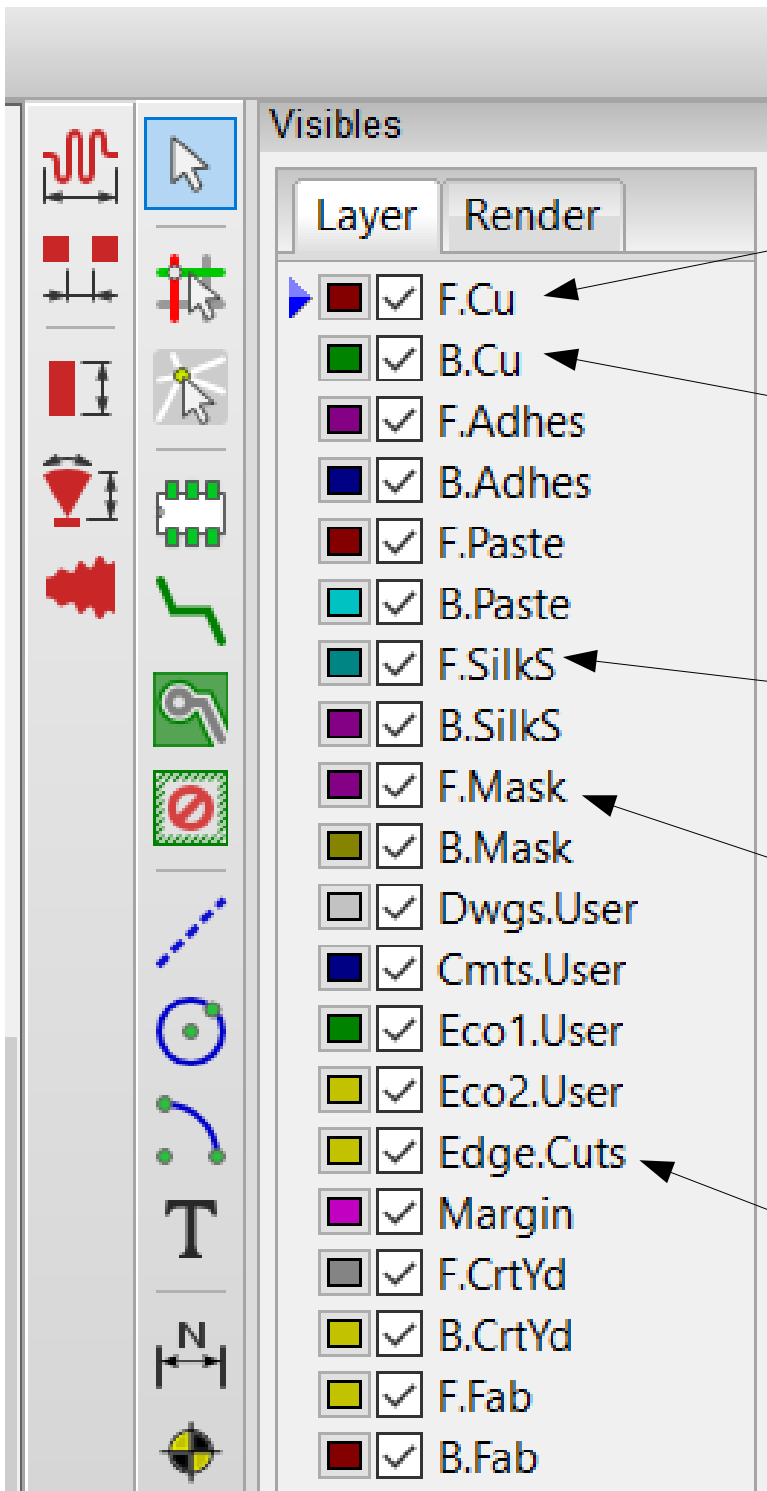


These rules determine via sizes and minimum feature sizes and widths.



These are the widths of the tracks that can be placed. 0.5 mm is placed here.

Commonly used board Layers



Front copper layer (where traces go on the front of the board)

Back copper layer (where traces go on the back of the board)

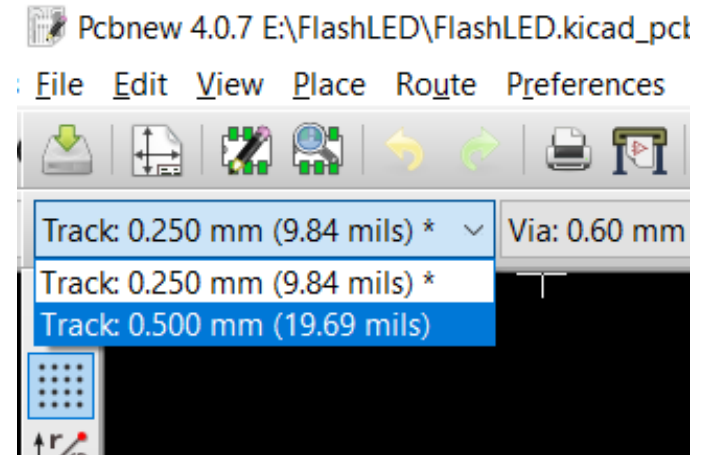
Front silkscreening layer, so you can annotate your PCB

Front and back solder mask.

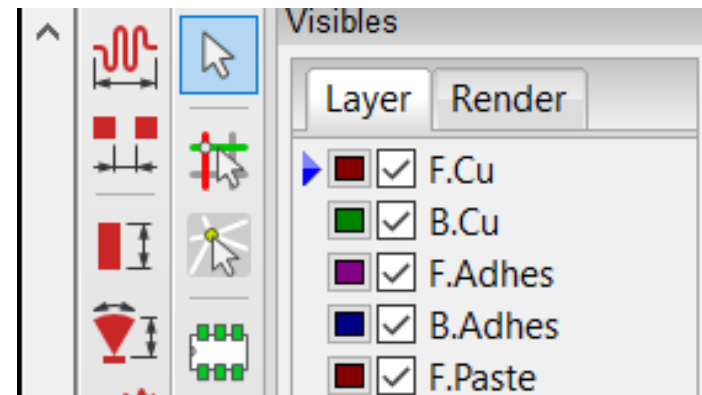
Edge Cuts (not used by all manufacturers), denotes the boundaries of your PCB (usually a rectangle)

Drawing the traces between components

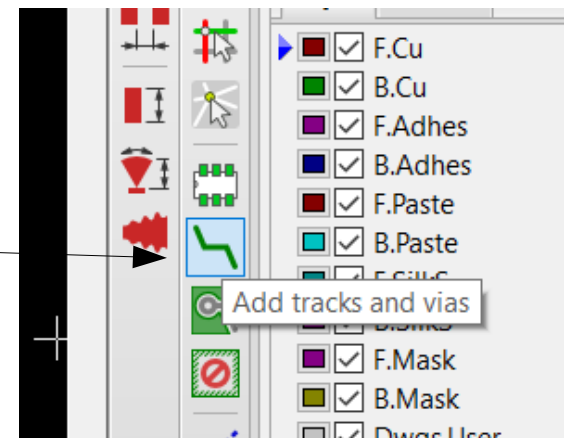
We move and rotate the components into position. To draw a trace between two pads, we first select the trace size:



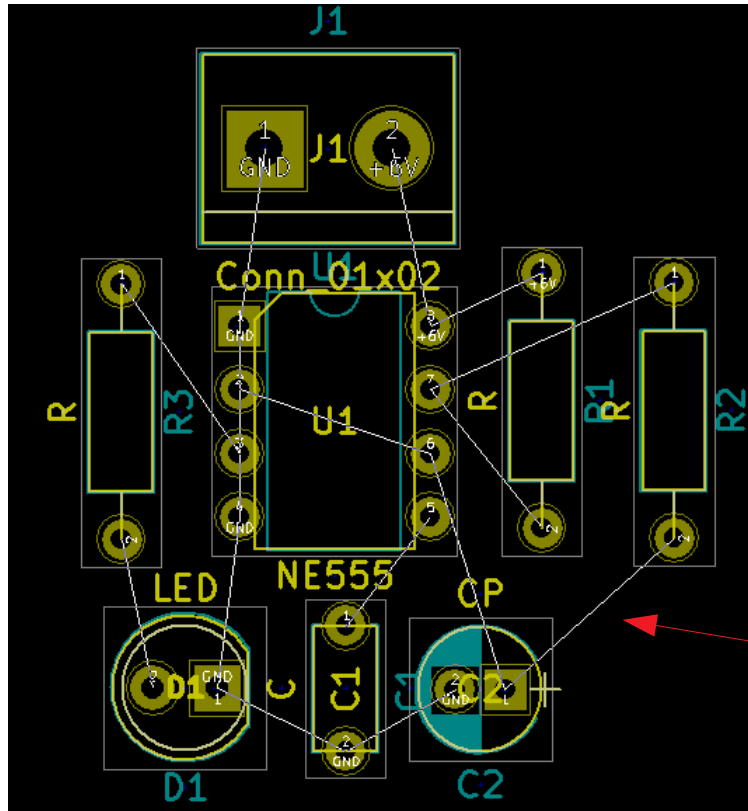
Select the layer the trace goes on by clicking on the layer name. The arrow appears next to the selected layer.



Then we select the draw track tool.



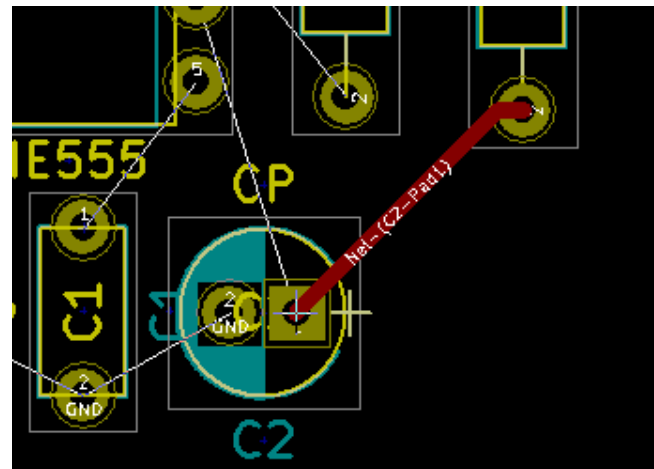
Connecting the components



The ratsnest tells us which components to connect. Kicad will not allow us to make connections not in the ratsnest, as these are not compatible with the schematic.

Pick this connection

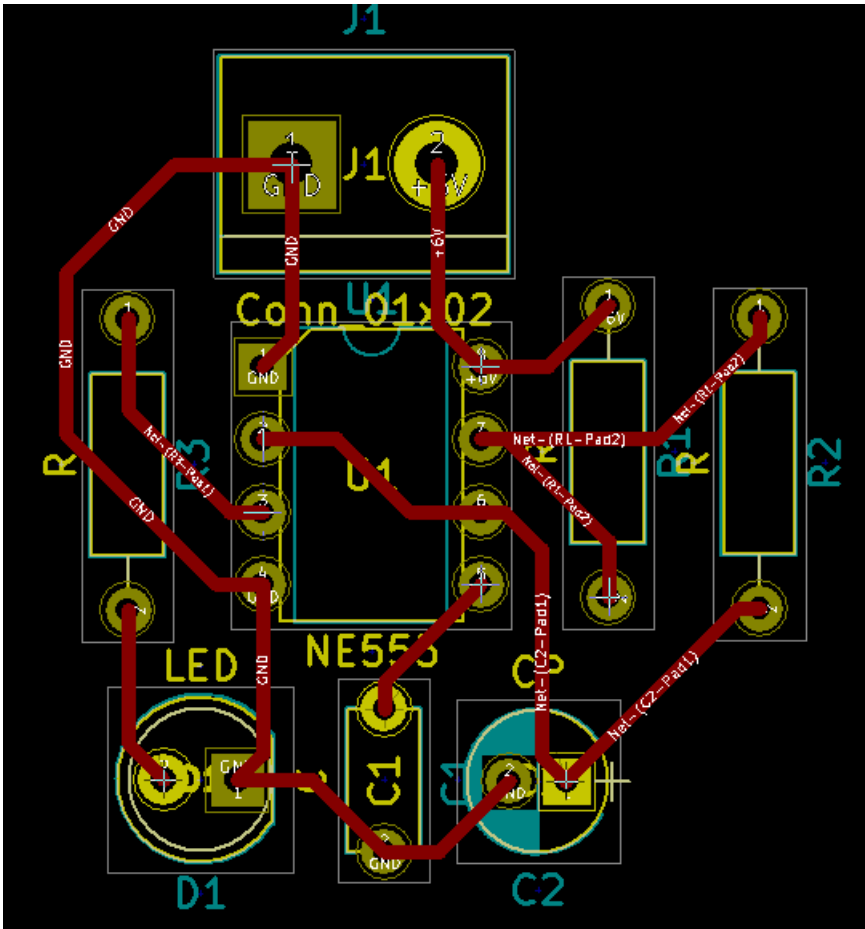
Click on one of the ends of the ratsnest, and guide the trace to the other end, and click the other end.



The ratsnest line is replaced by a trace with the color given by the layer.

Filling in all of the traces

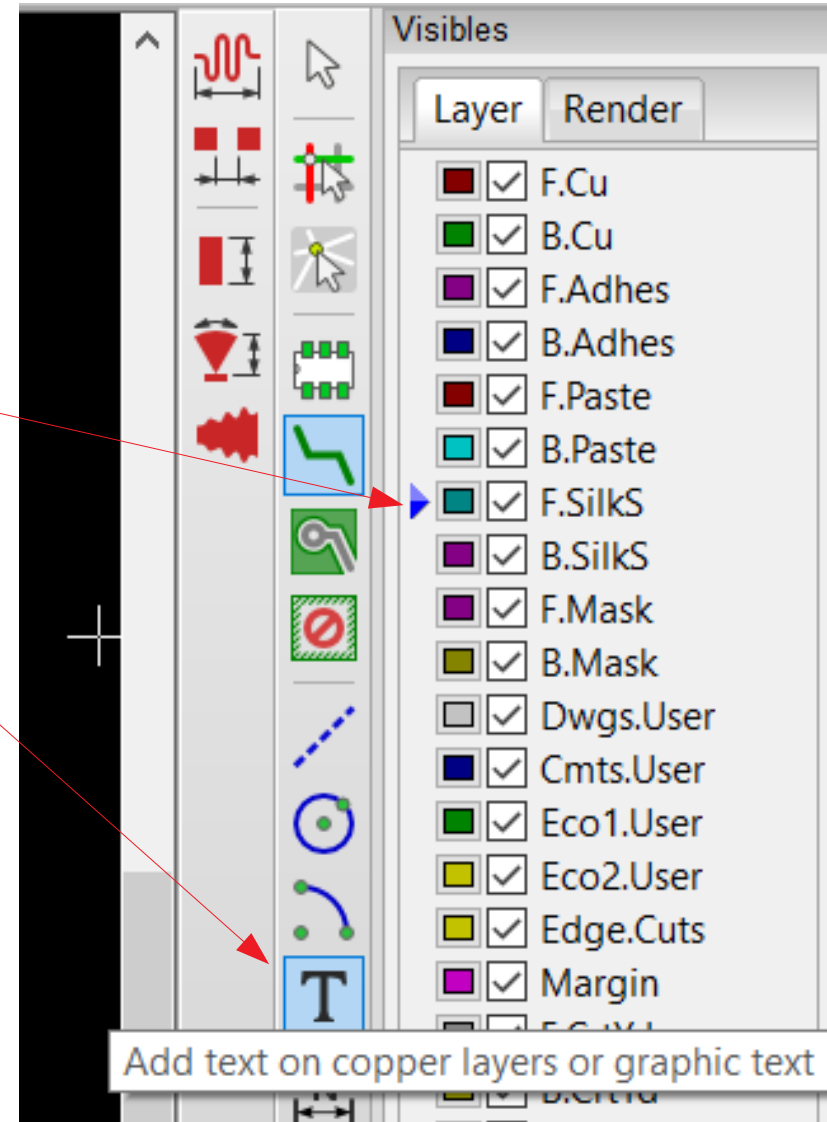
We can add silkscreening



Each trace is labeled by its “net name” which describes which pads it connects.

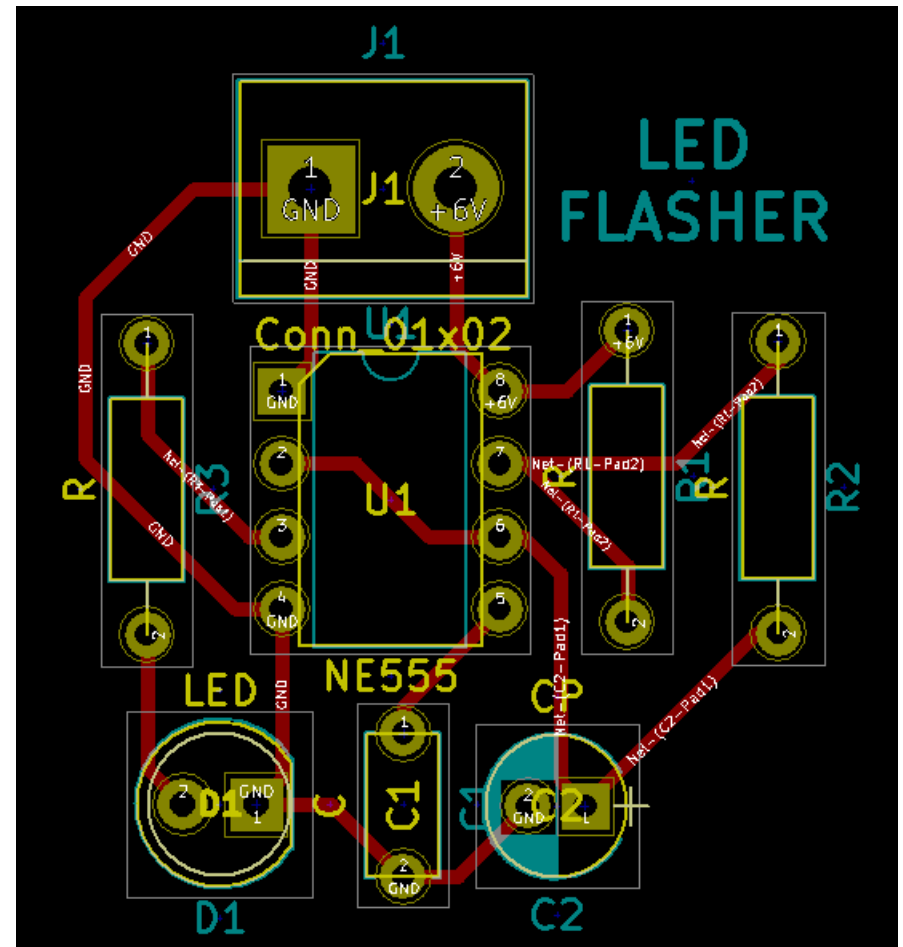
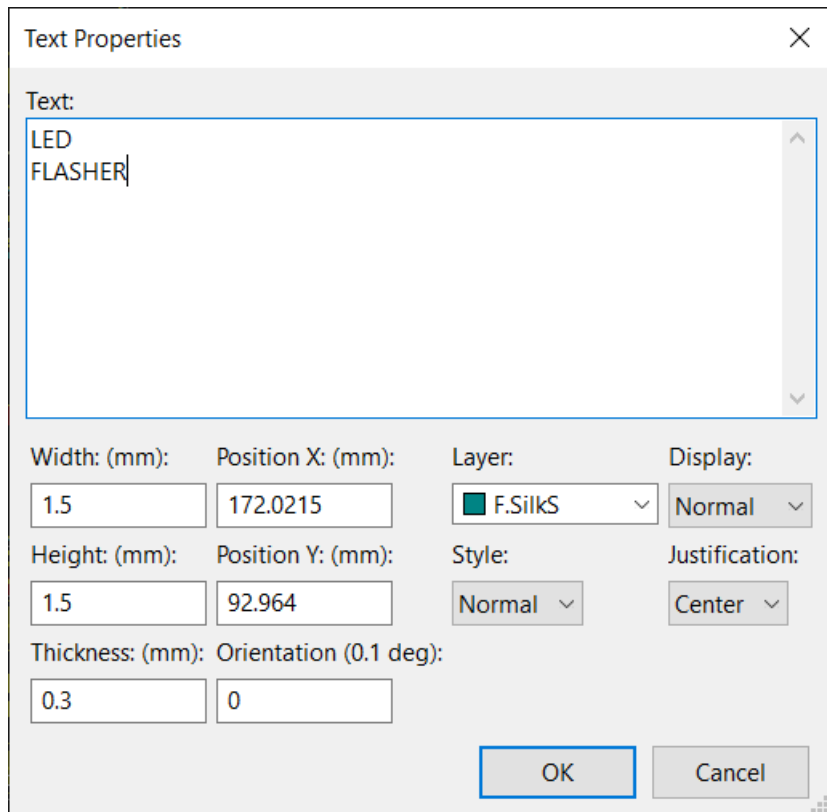
Select silkscreen layer

Text tool



Adding silkscreening

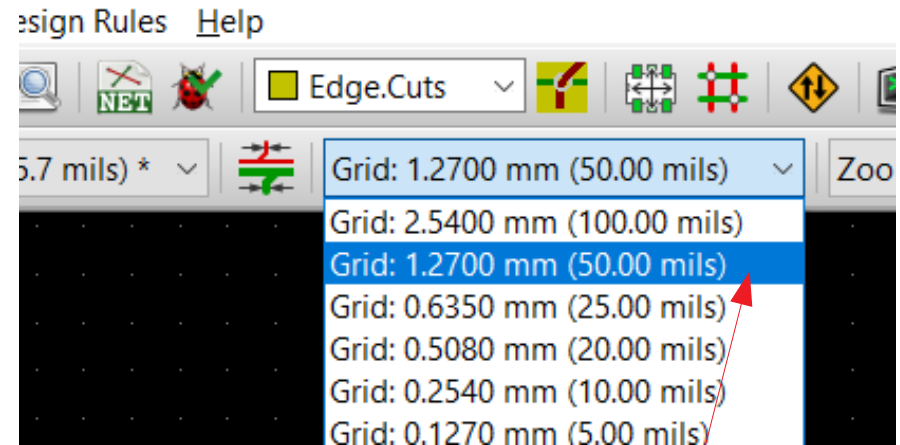
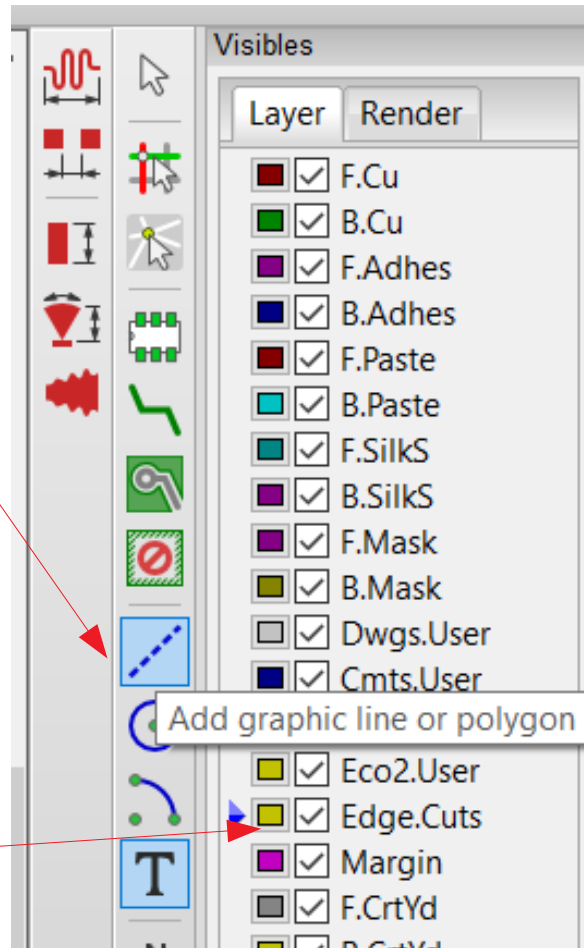
Click on the PCB where you want to add the silkscreening and then type the text into the box:



Drawing the board outline

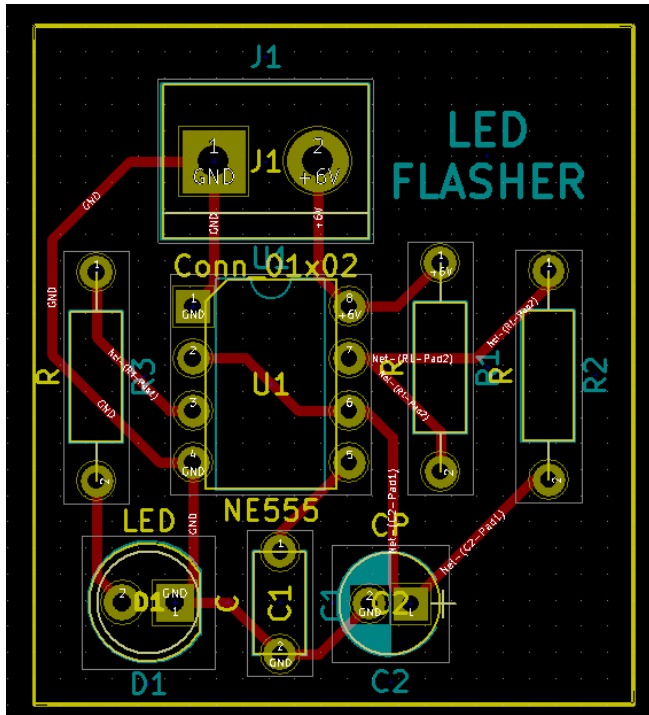
Choose line tool

Select edge cuts layer

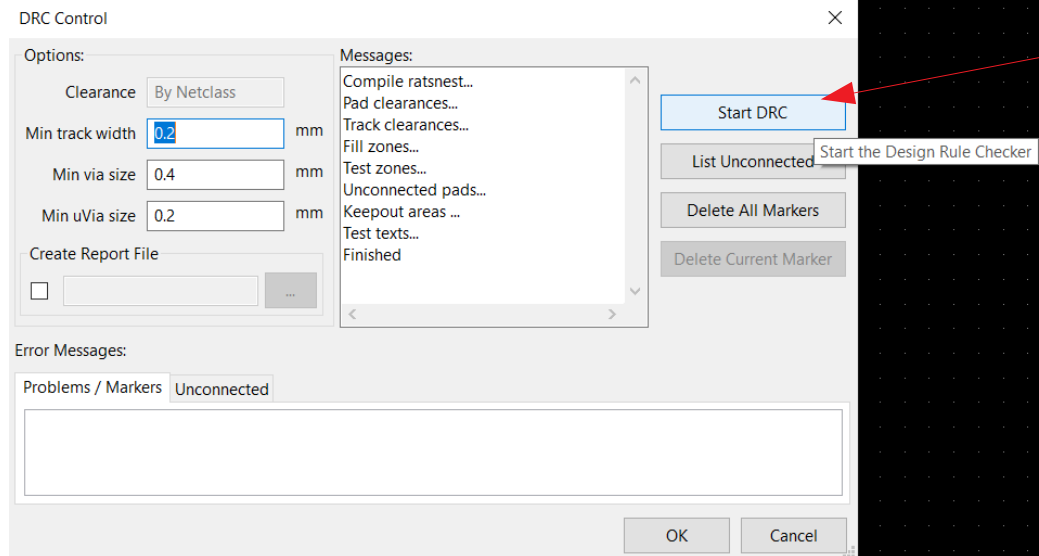
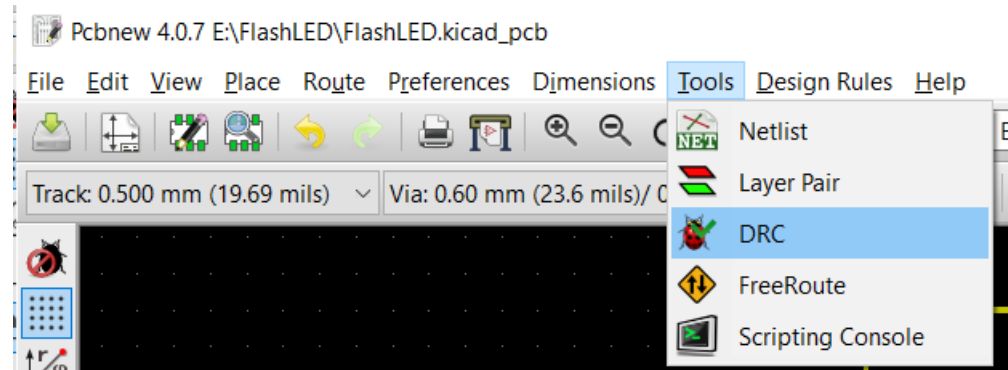


Select a coarse grid size to make it easier to draw a straight rectangle

With the board outline

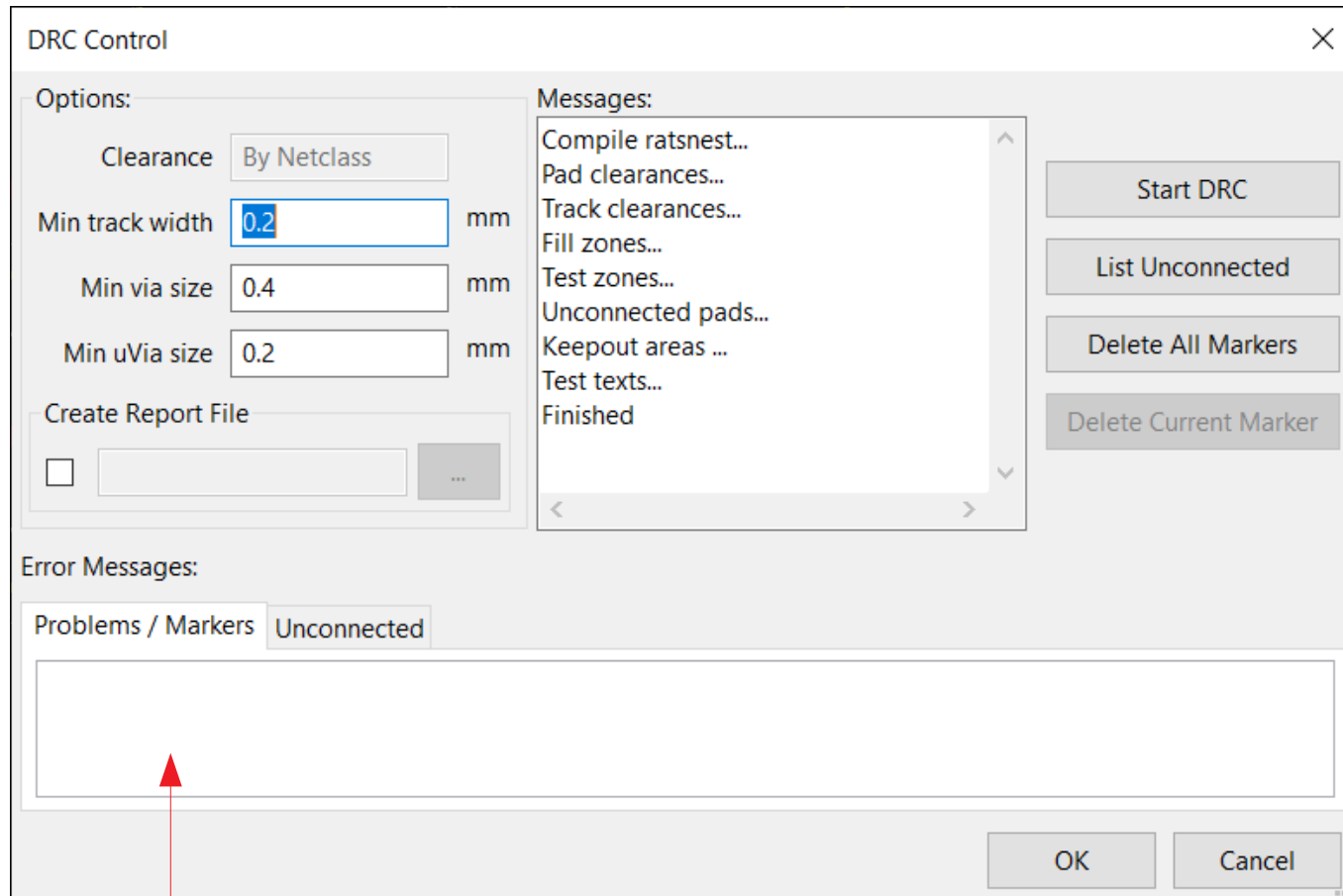


Now we do a design rules check to make sure everything is ok!



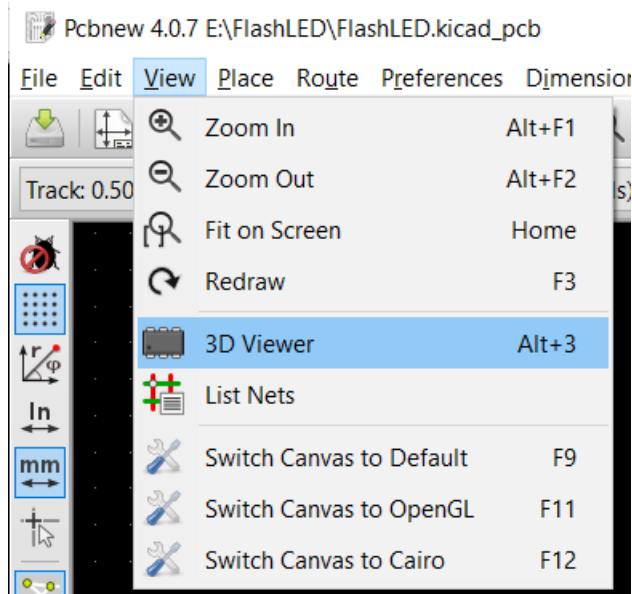
Do design rule check

Design rule check success!

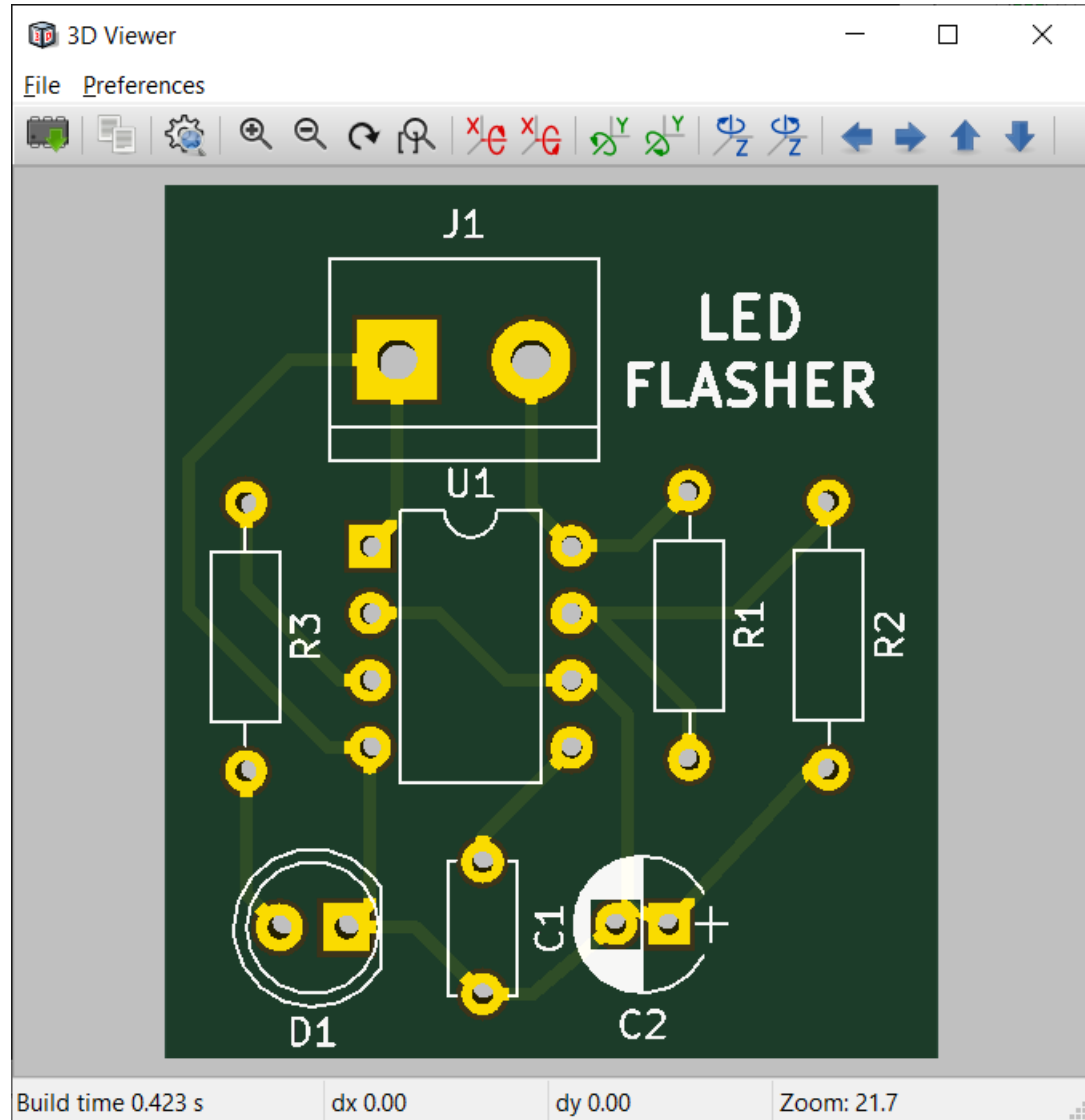


Problems can be parts are too close together or touching that should not be, too small width between traces, etc. If you click on the problem it takes you the location of the PCB.

What does my PCB look like?



Use the 3-D viewer

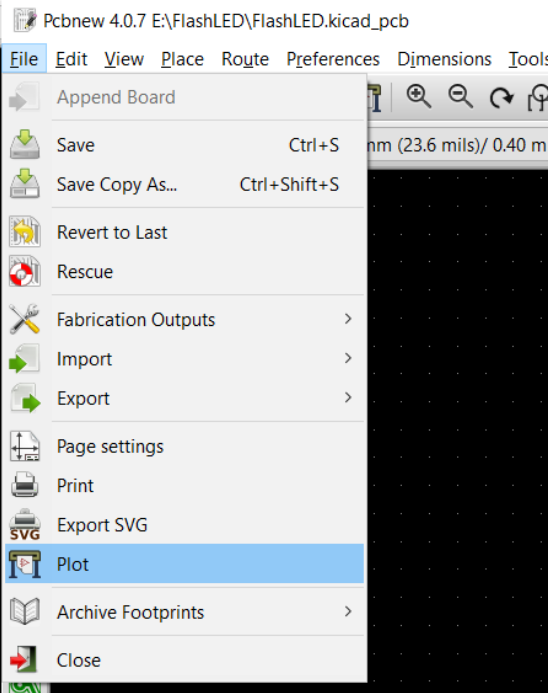


Exporting the Gerber files

Gerber files are what you send the PCB manufacturer to make your PCB.

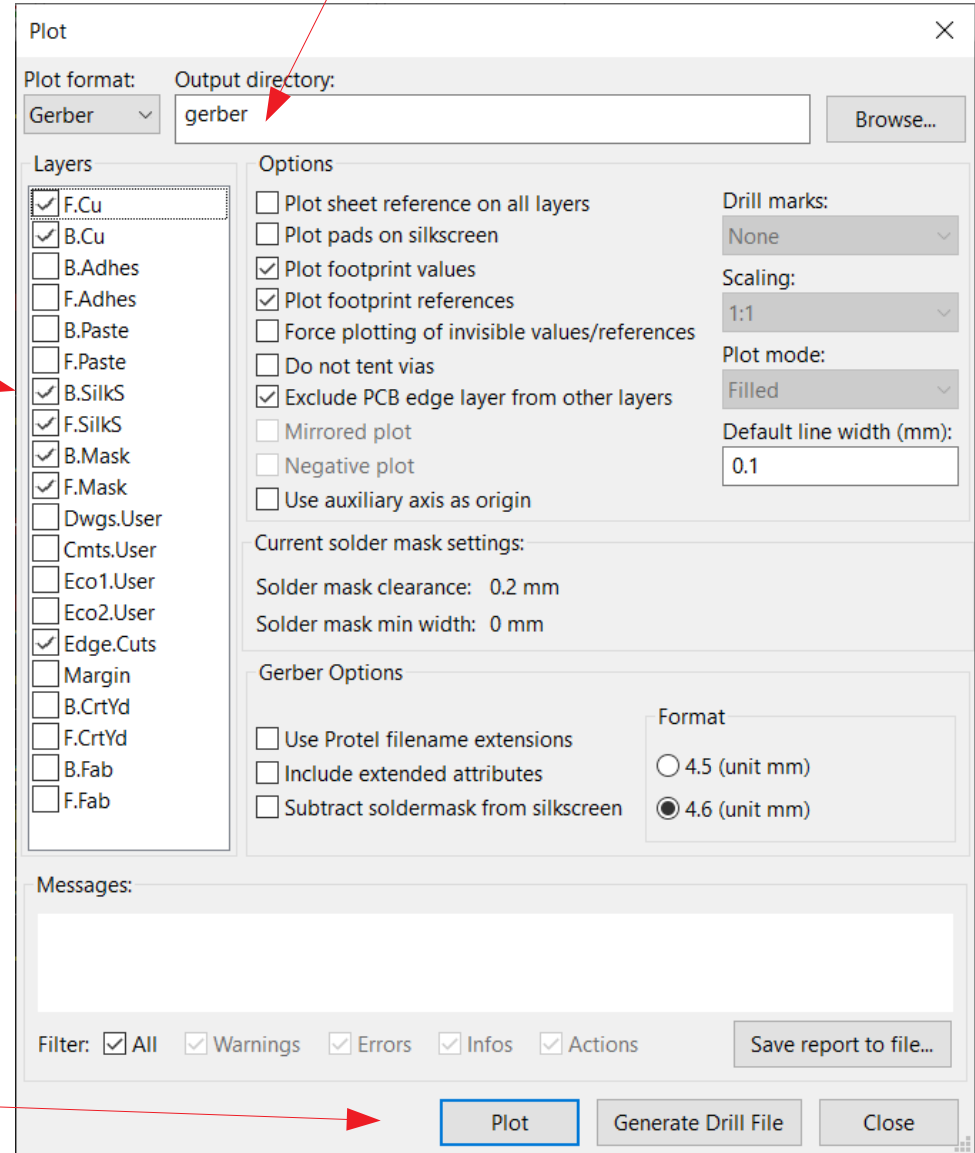
Enter “gerber” here so the files are in a separate directory.

Check these layers, which are typically what is needed (unless you need a solder stencil as well)



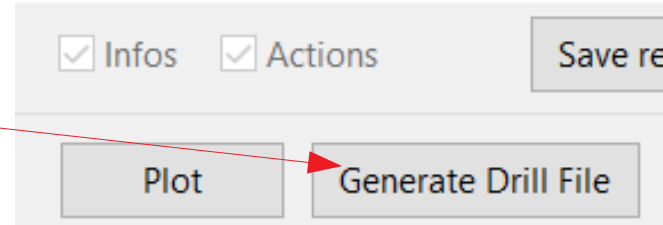
Select plot from the menu

Click “plot” to generate the gerbers

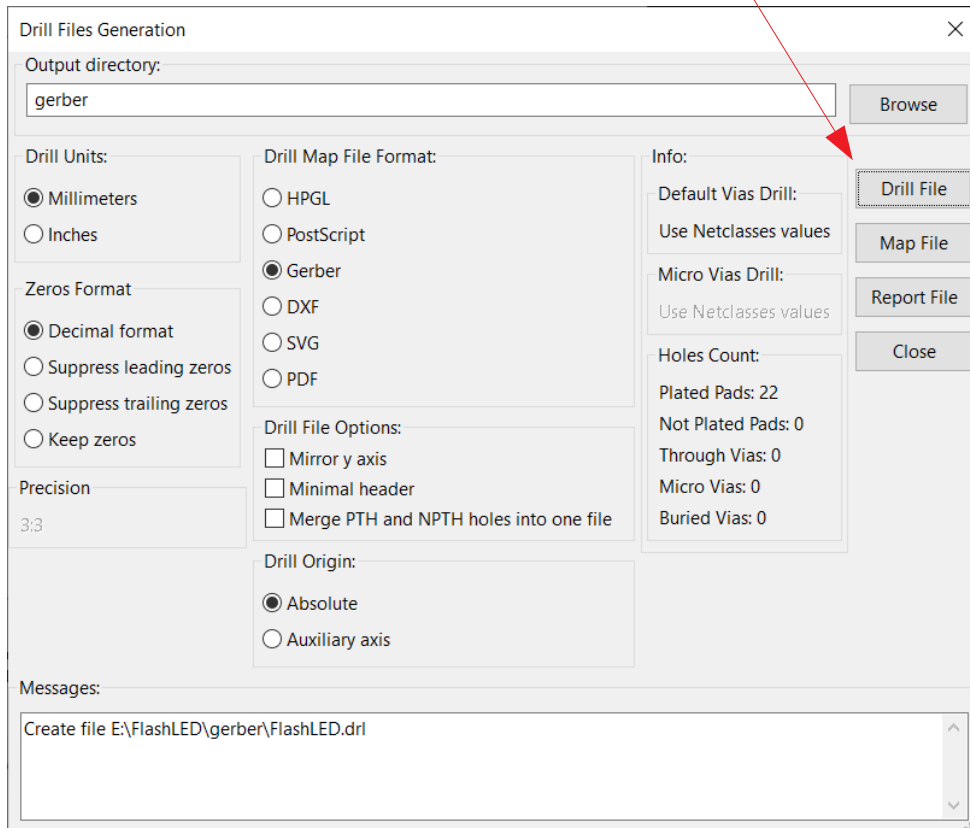


Exporting the Excellon Drill Files

You also need files to indicate the holes to drill.



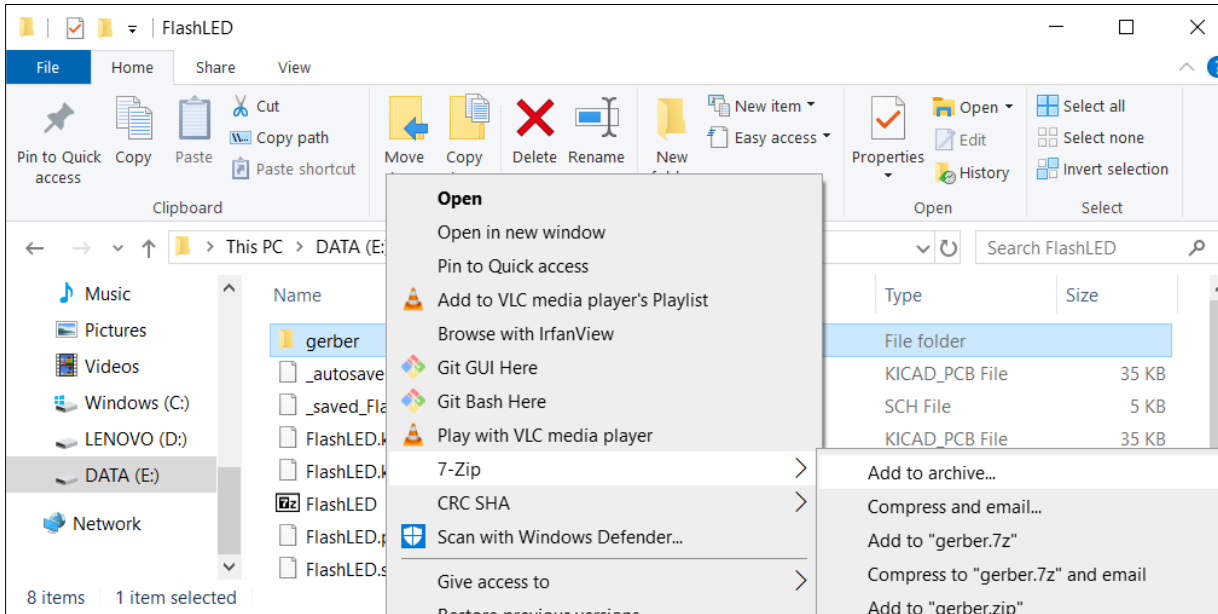
Click here to generate drill files.



The drill files go into the same directory as the gerber files.

Sending your job to be manufactured (example, JLCPCB)

We zip up the gerber directory into a single file



Use your favorite zip program (I use 7-zip under Windows)

Now we have a zip file of the “gerber” directory.

Log onto JLCPCB website (www.jlcpcb.com)

PCB Prototype SMT Stencil

GET INSTANT QUOTE

Dimensions: 100 X 100 mm

Quantity: Choose Num (5pcs)

Layers: 2 Layers

Thickness: 1.6 mm

QUOTE NOW

Click “quote now.”

Click “add your gerber file.”



Add your gerber file

Only accept zip or rar,Max 4 M

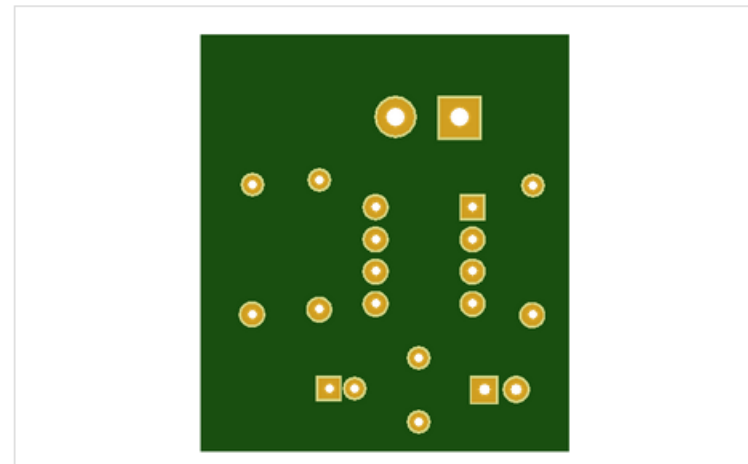
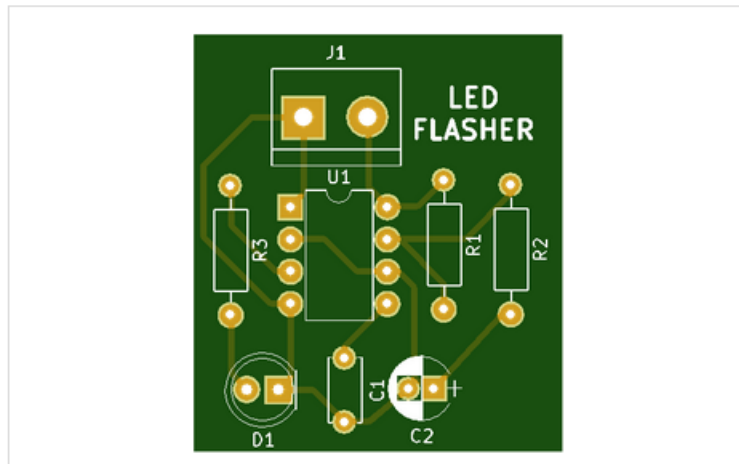
[How To Generate Gerber Files](#)

Find and upload the gerber zip file.

And here it is, you can see the front and back of the PCB board in the on-line viewer.

Detected 2 layer board of 33x29mm(1.3x1.15 inches) .

Your upload has finished processing. Enter the project details below and we'll move on to checking all the individual layers to make sure that they're correct.



The gerber viewer is for reference purpose only and may differ from the actual PCB product.

[Gerber Viewer](#) 



Use the “Gerber Viewer” link to check to make sure JLCPCB’s system is reading your gerber files correctly.

Finish filling out the form and complete the order.

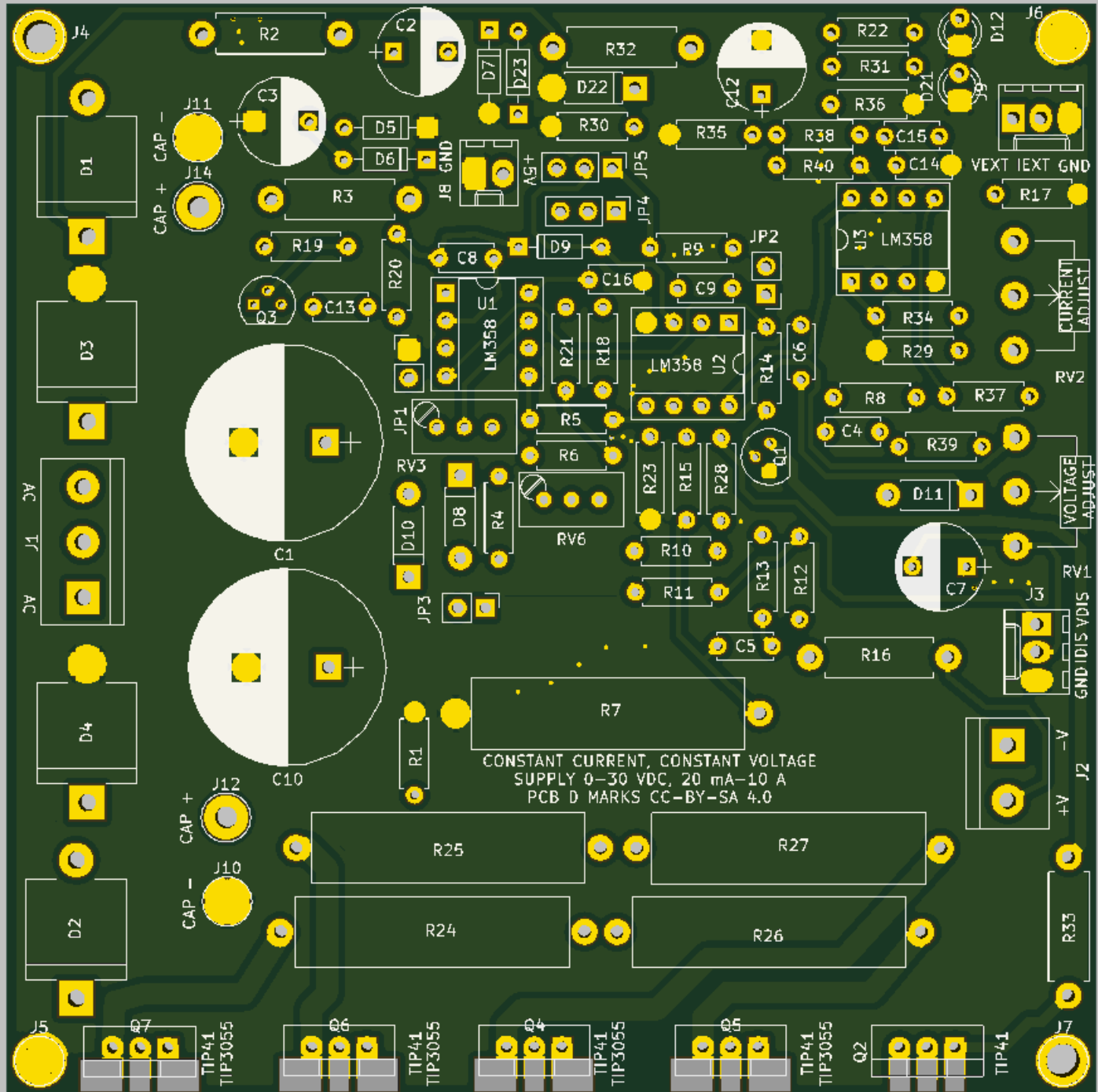
And that is an example on how to design a PCB with Kicad and have it made, start-to-finish!

Designing library symbols and footprints are important tasks, but not covered here.

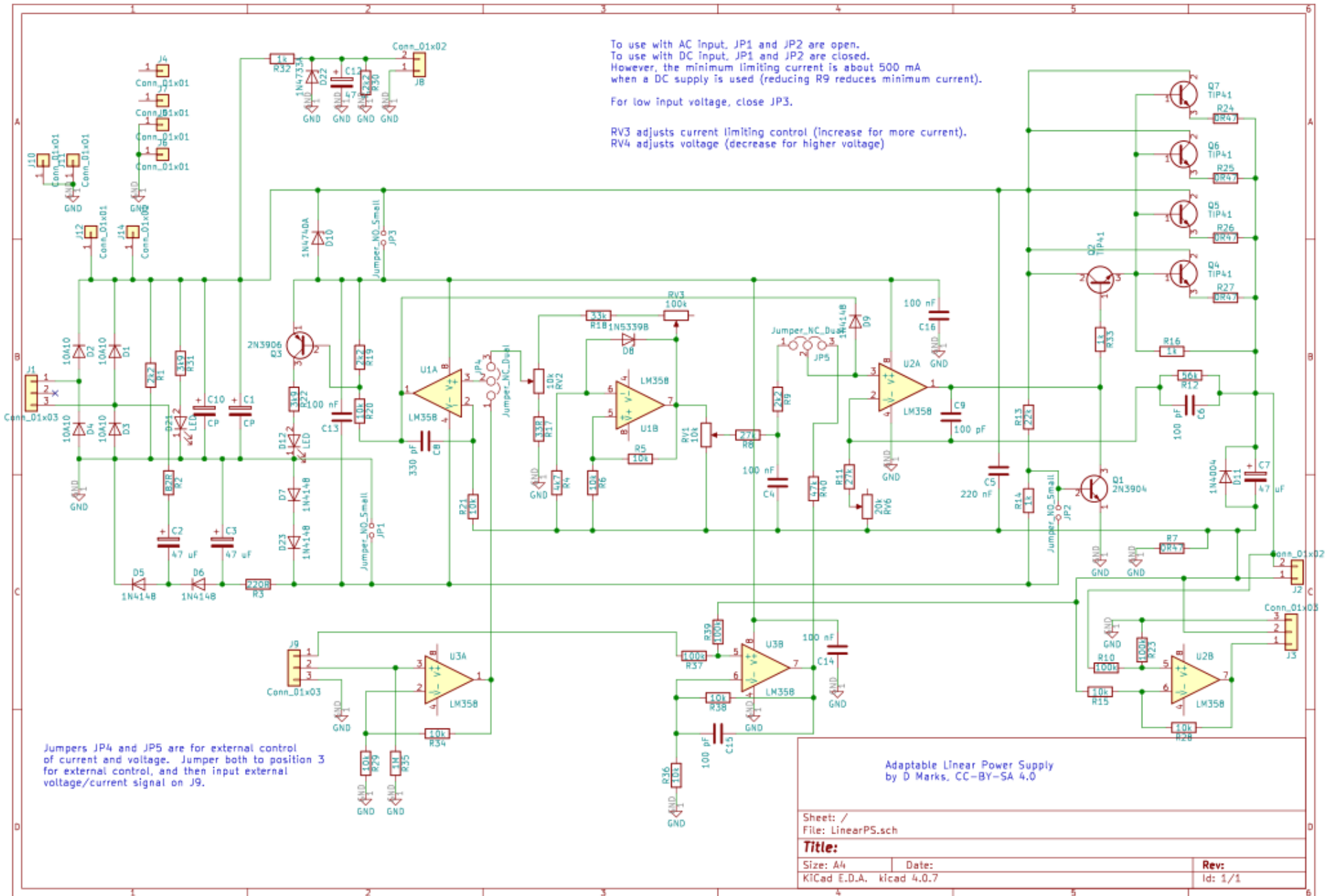
Also adding copper fills and vias are not covered in PCB layout, also important.

Some examples:

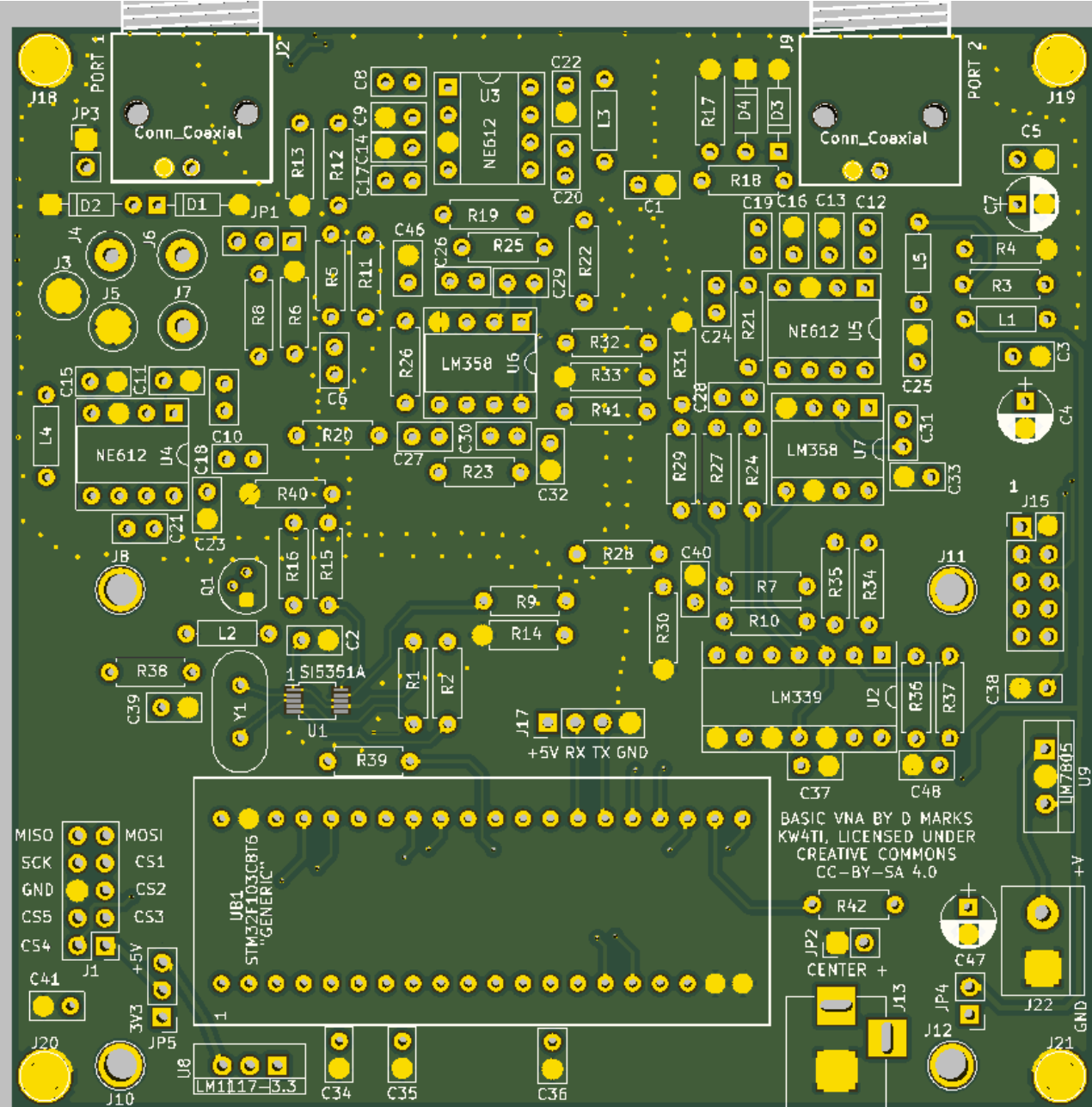
Power Supply PCB



Power Supply Schematic



Some examples: VNA PCB



Conclusions

Hams across the world work together on open source projects that are freely shared by all.

If you have an idea, go for it! Open-source tools like Kicad are out there to be used by people like you.

If you like a project, participate! Be a user, document something, make a instructional video, or even design a circuit or write some code.

Kicad is available at www.kicad-pcb.org

My projects are available on

<http://www.github.com/profdc9/>