KICAD JUMPSTART

INTRODUCTION TO ELECTRONIC PRINTED CIRCUIT BOARD DESIGN USING THE KICAD ELECTRONICS DESIGN SUITE

WARREN MERKEL, KD4Z
FEBRUARY 2022

KICAD – PRINTED CIRCUIT BOARD DESIGN SOFTWARE

- KiCad is a suite of programs that give the user powerful tools to draw electronic circuits in schematic form, and create printed circuit layouts from them.
- KiCad was started in 1992 by a French university professor needing a teaching tool.
- KiCad is Open Source, and can be used on Windows, Linux or macOS platforms.
- KiCad is FREE!

TERMINOLOGY

- First, we need some definitions used in the Printed Circuit world.

- Through-hole (TH) vs. Surface Mount Devices (SMD)

- Symbols and Footprints

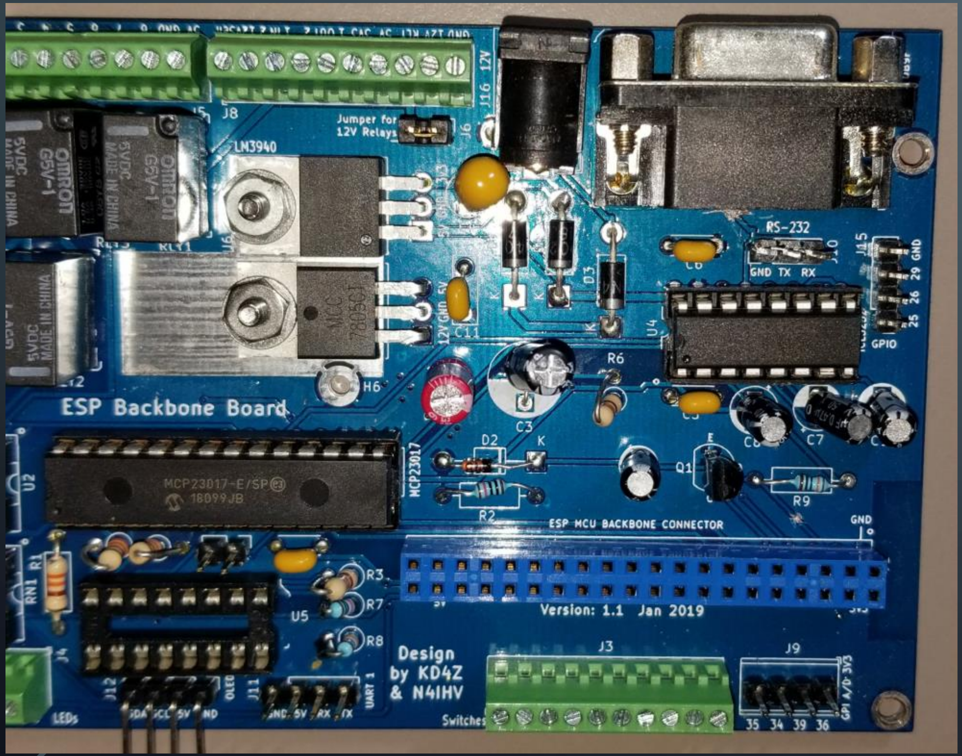
- Then, we need a circuit design to do something useful.

BOARD CONSTRUCTION

- Typical PCBs have two or more layers of copper separated by fiberglass resin.
- Two sided boards (or 2 layer) are commonly used, though with higher density pin counts found in CPU / FPGA devices, more layers are required just to gain access to all of the pins.
- Computer motherboards typically have 8 to 10 copper layers.
- Extra layers are commonly used as ground shields between signal layers, and for power rail distribution.

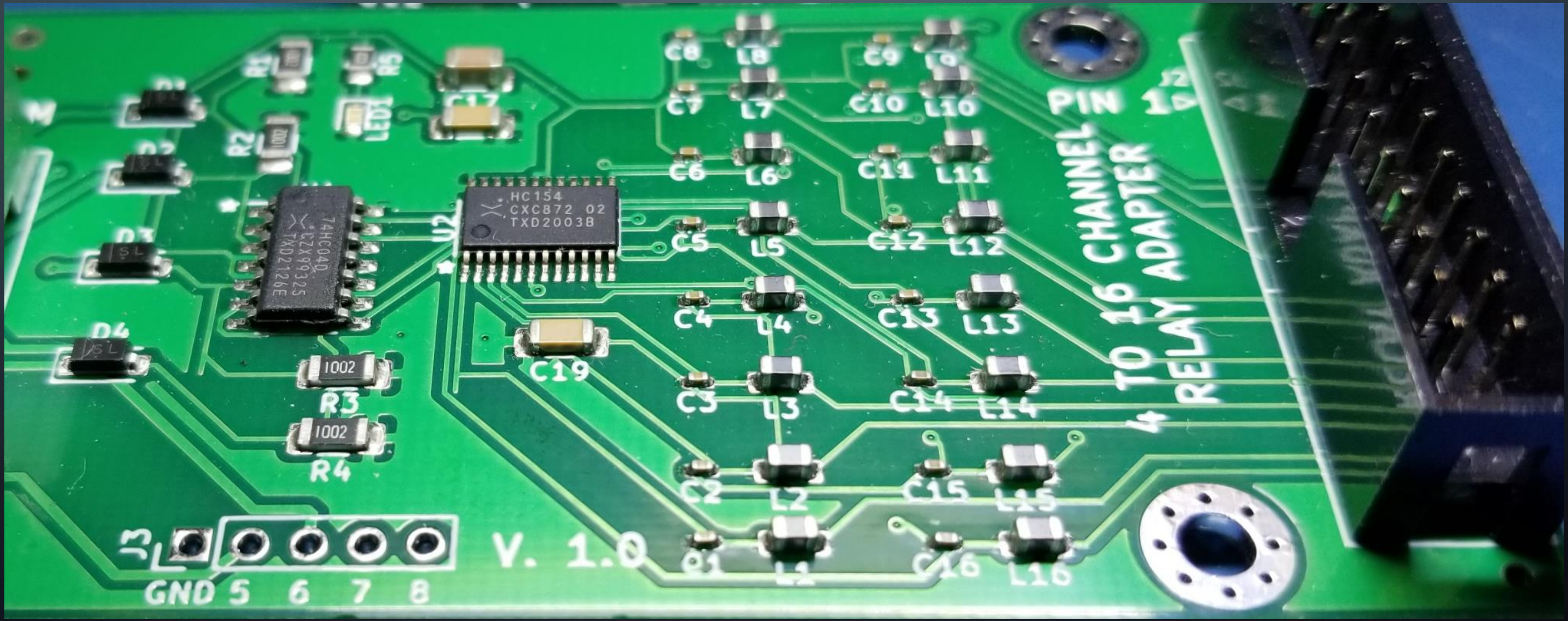
SOME DEFINITIONS

- Through-hole – Holes are drilled through the circuit board, in a size to accept the component leads.   Multilayer boards will have copper plating inside the hole, connecting each desired layer.

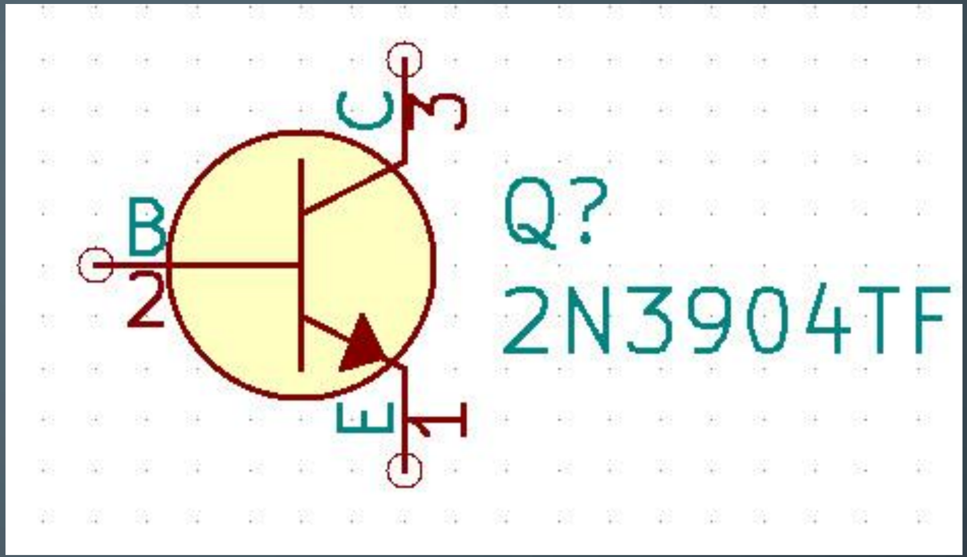


SOME DEFINITIONS

- Surface Mount – No holes for components. Components are soldered to pads on the board. Traces change layers using vias, which are small plated through holes.

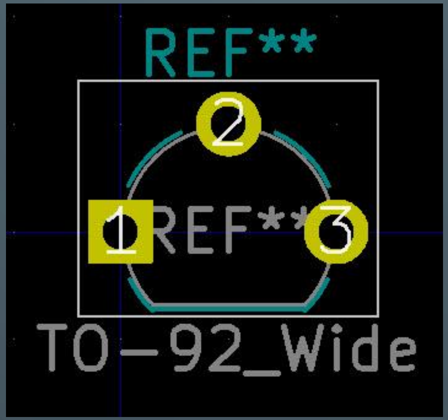


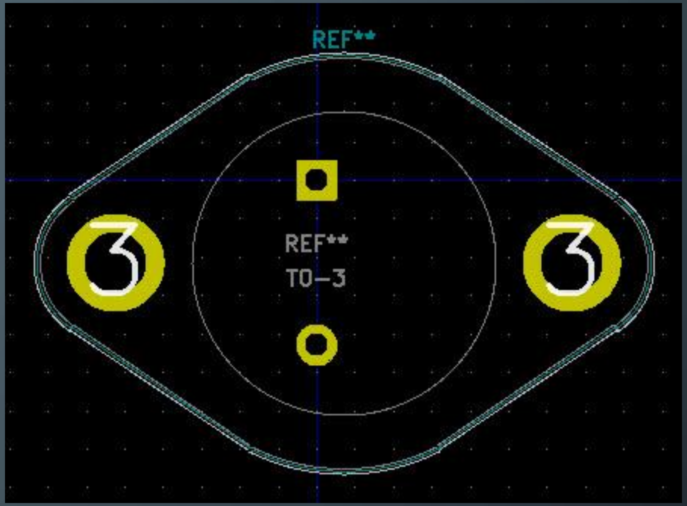
SOME DEFINTIONS – SYMBOLS



- This is a **Symbol** for an NPN transistor.   The schematic will be drawn with symbols representing the parts you want to use. Resistors, Capacitors, Inductors, Semiconductors, all have recognizable symbols when drawn on a schematic diagram.
- It represents the function of the part, but not the physical design or package of the part.

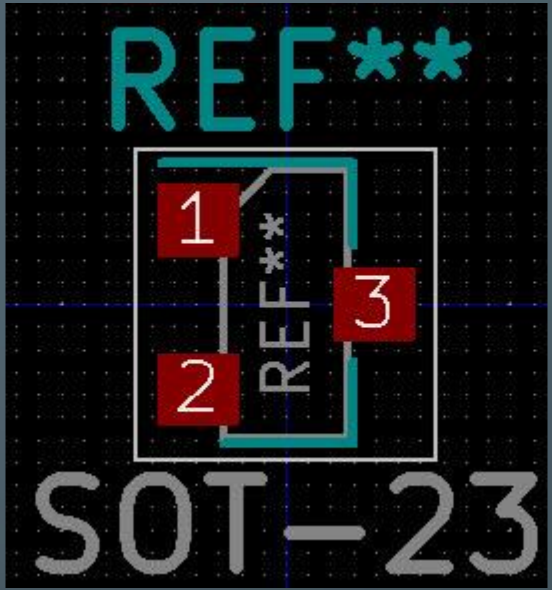
SOME DEFINTIONS – THROUGH HOLE FOOTPRINTS

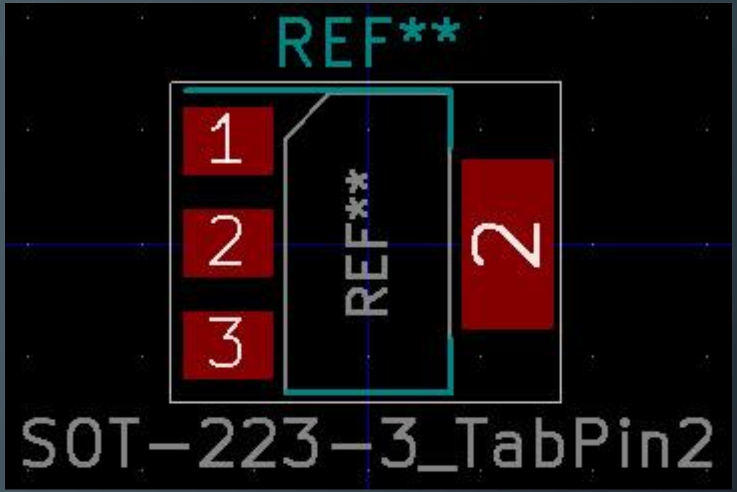




- These are all Through-hole **Footprints** for the same transistor **Symbol.** Each connection requires a tiny hole drilled through the board.
- It represents the physical design of the part and what it needs for connections on the printed circuit board. Footprints are always "TOP LOOKING DOWN".
- **Symbols** are associated with a specific **Footprint.** Though, you can have many similar symbols, each with a specific footprint depending on the package.

SOME DEFINTIONS – SURFACE MOUNT





- These are Surface Mount Device (SMD) **Footprints** for the same transistor **Symbol.**
- It represents the physical design of the part and what it needs for connections on <u>one side</u> of the printed circuit board only. Surface mount parts <u>do not have holes</u> for the Pin connections.
- Surface Mount parts are typically much smaller than Through-hole devices. That is a benefit in RF designs as the extra capacitance due to the connection leads is minimized.

PICK A PROJECT AND DESIGN A BOARD!

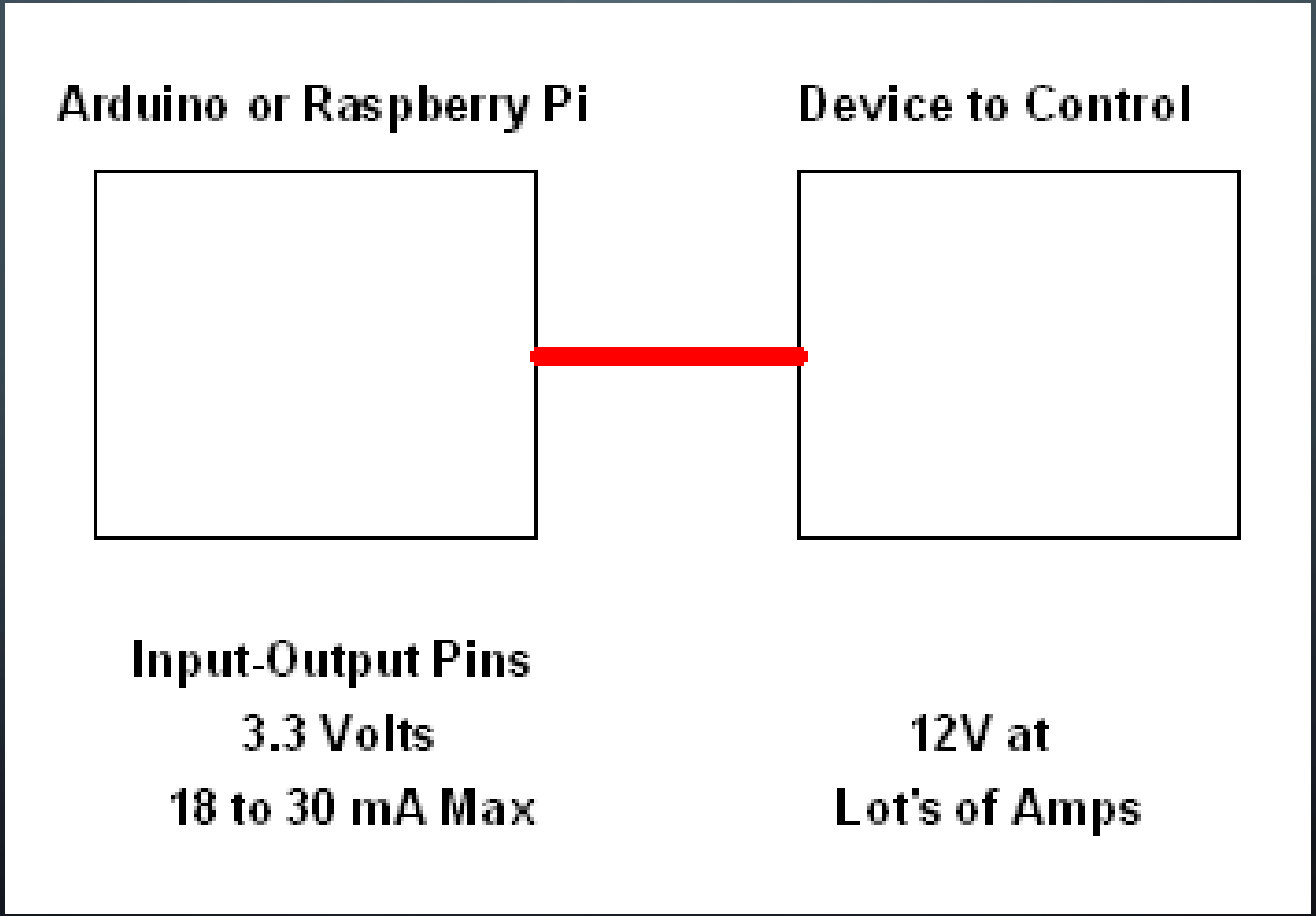
- For this presentation, let’s design something we might want to use in our shack.
- Lots of interest in IOT and Remote Control of devices.
- Design a power switching board that can handle the large current requirements of an 100 Watt HF radio, yet can be controlled by a low power device such as an Arduino or Raspberry Pi.

DEFINE THE CIRCUIT REQUIREMENTS

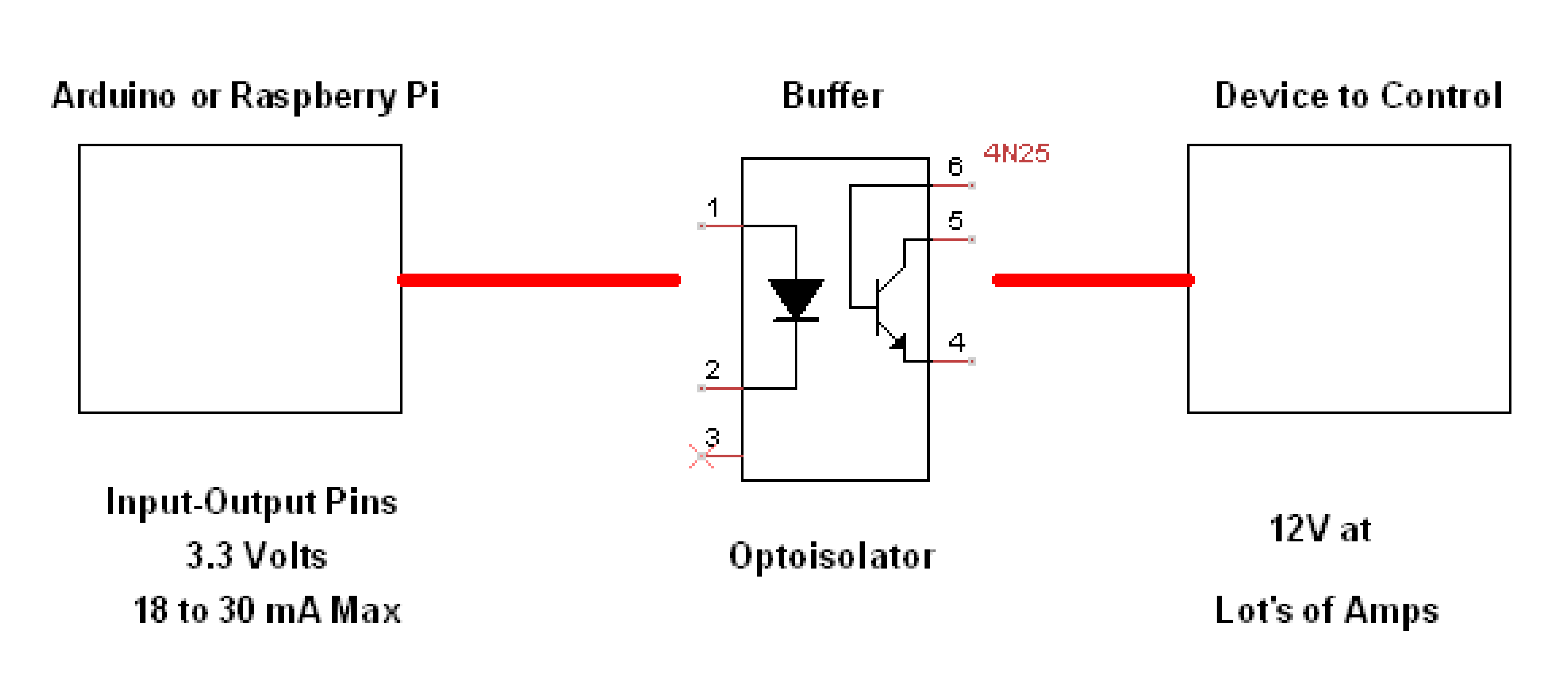
- Provide isolation between the IO pins and the device being controlled.
- Must be driven with a low voltage (Logic Level)
- Raspberry Pi / ESP32 IO pins are 3.3V, 17 to 40ma current sink/source maximum
- Arduino UNO/Nano IO pins are 5V, 20ma maximum sink/source.
- Typical Relays having large contacts require 100 mA or more for the coil.

Clearly, we can not drive a relay directly from an IO pin!

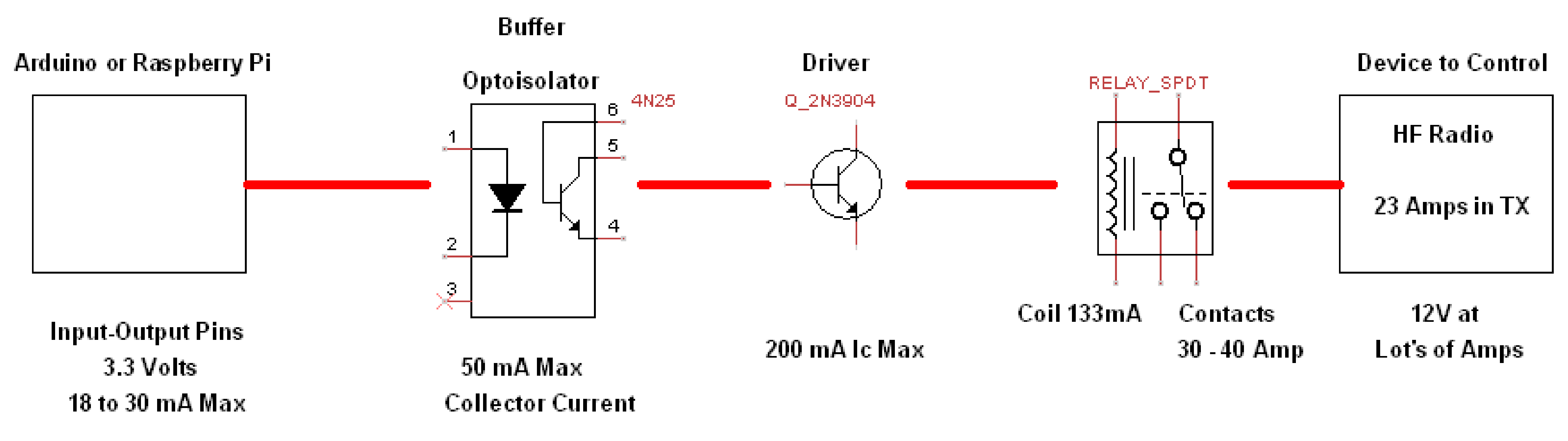
POWER SWITCHER CONCEPT



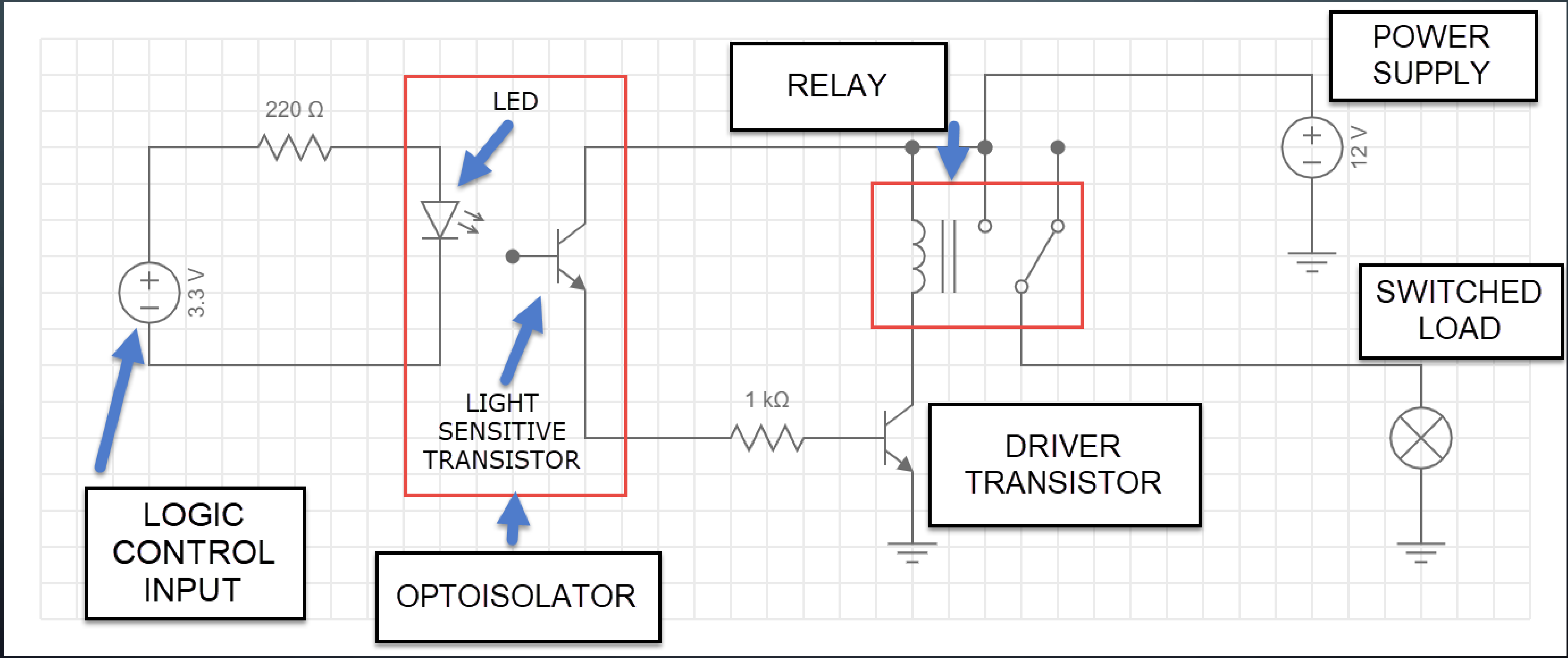
POWER SWITCHER CONCEPT



POWER SWITCHER CONCEPT

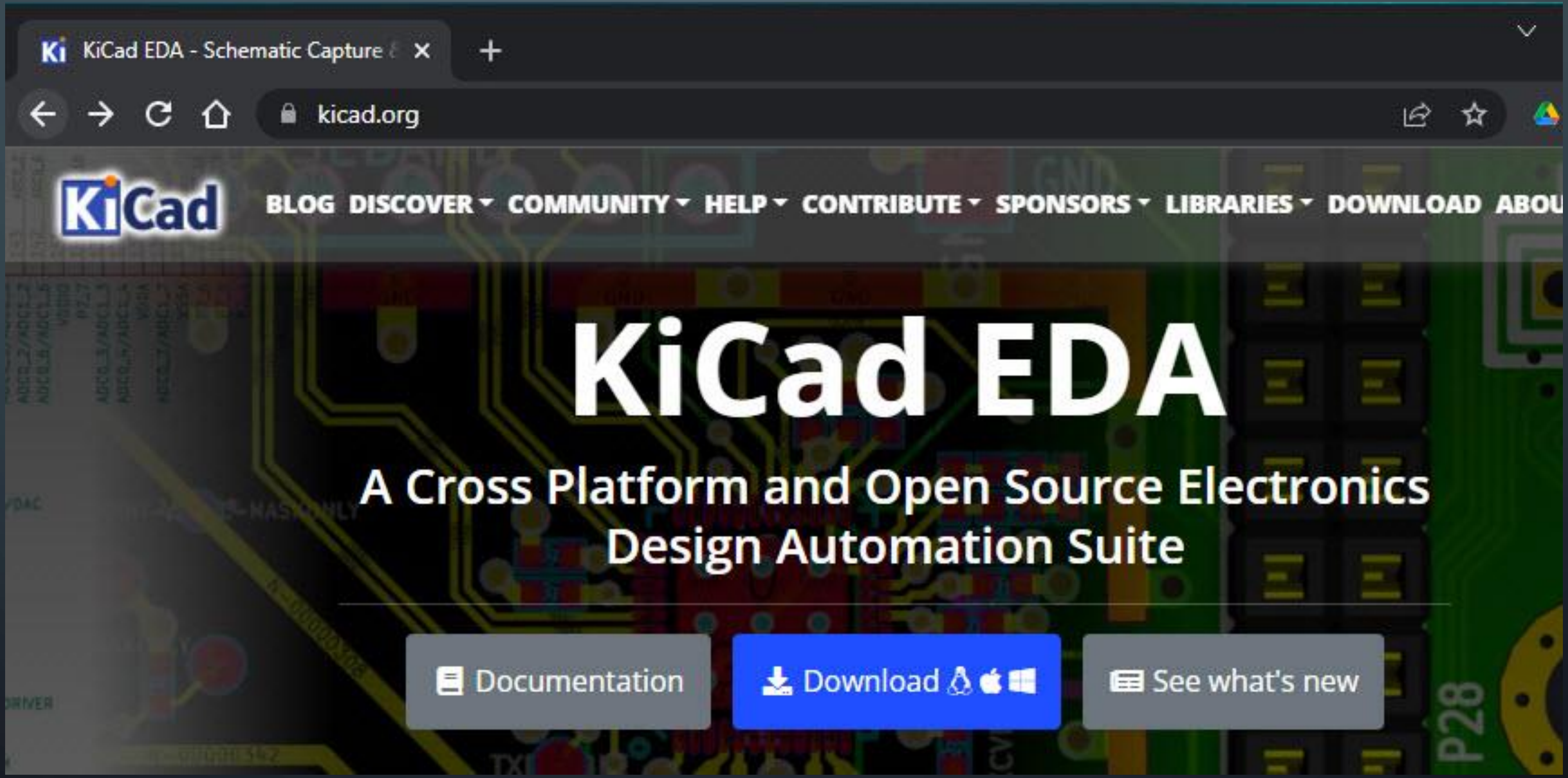


GET OUT THE GRAPH PAPER AND GET CREATIVE



GET STARTED

- Download and install the KiCad installer appropriate for your Operating System from https:\\kicad.org



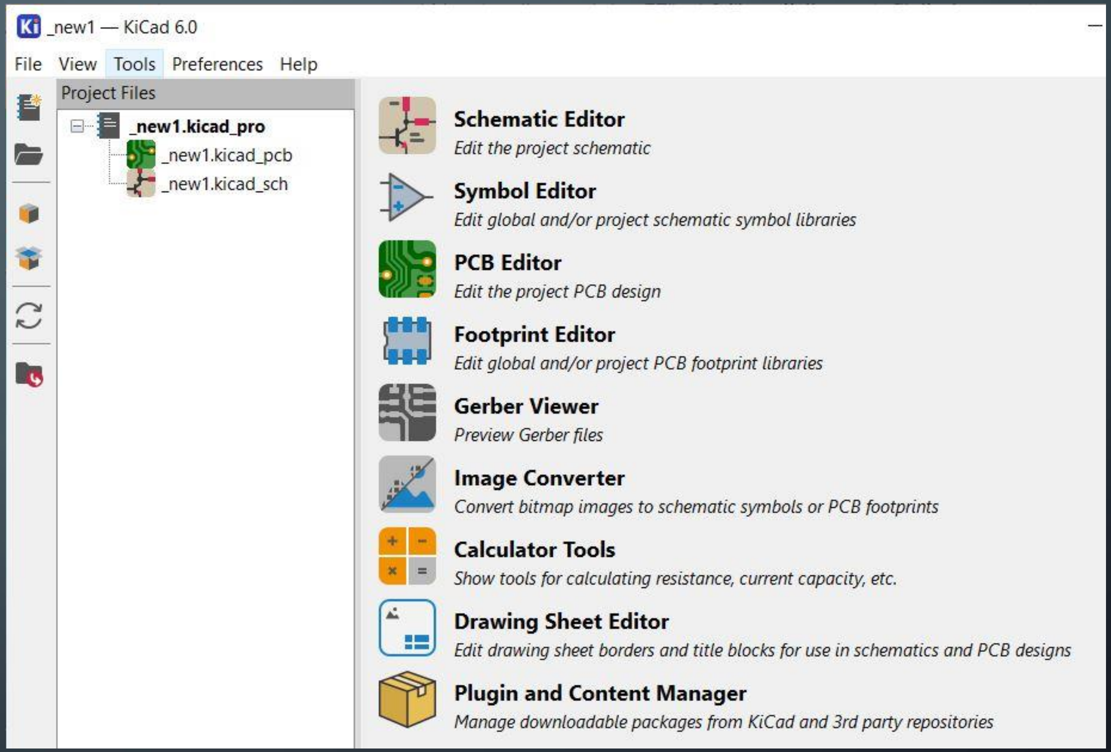
AFTER THE INSTALLATION IS COMPLETE

- The KiCad Suite has multiple programs that work together.
- You can access them from the KiCad project manager application.
- Find this Icon and launch it.

KICAD PROJECT MANAGER

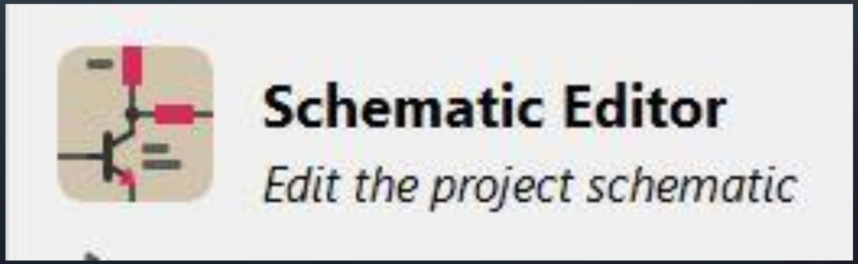
Each Icon represents a separate program or module of KiCad.

You will use the Schematic Editor and the PCB Editor the most.

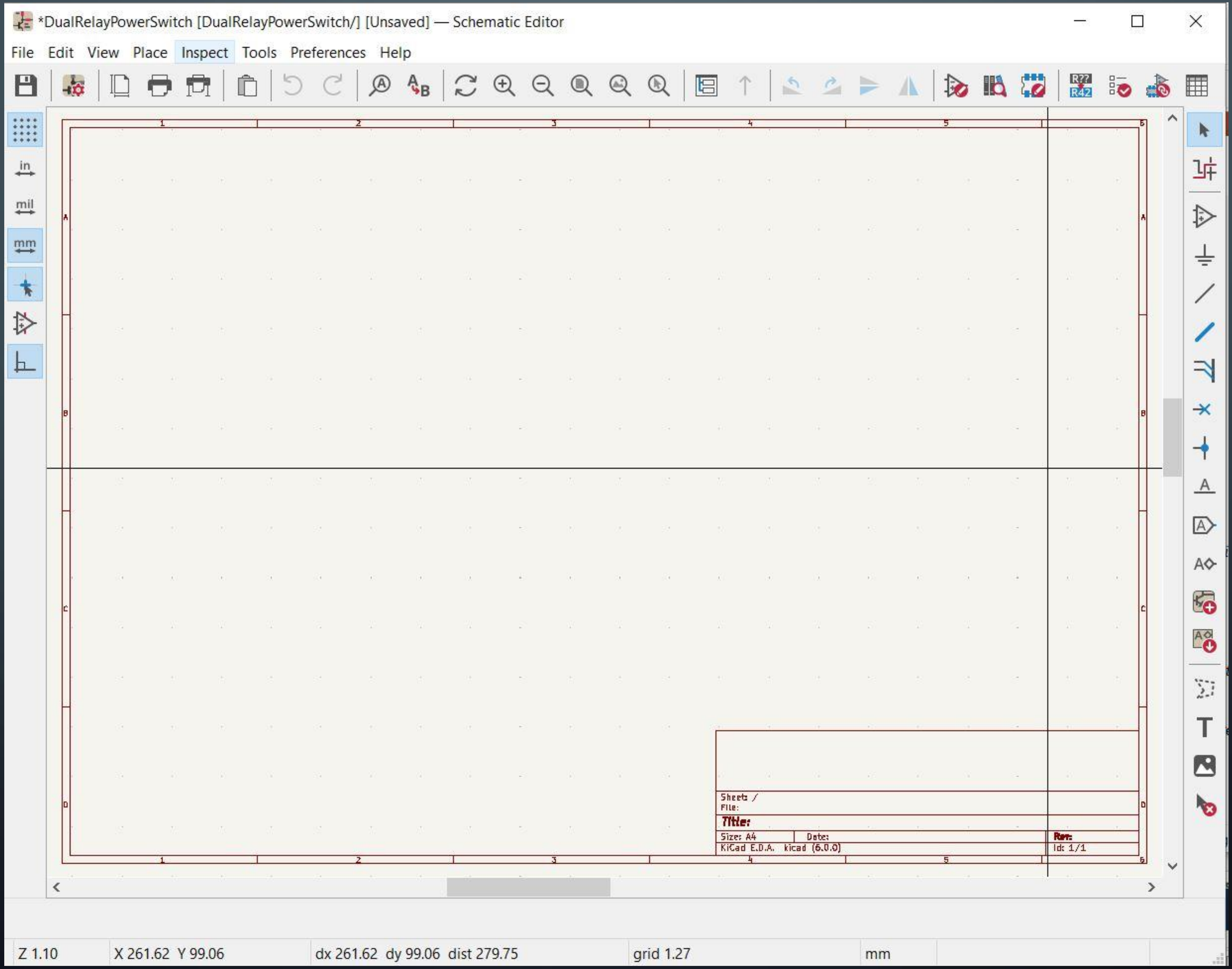


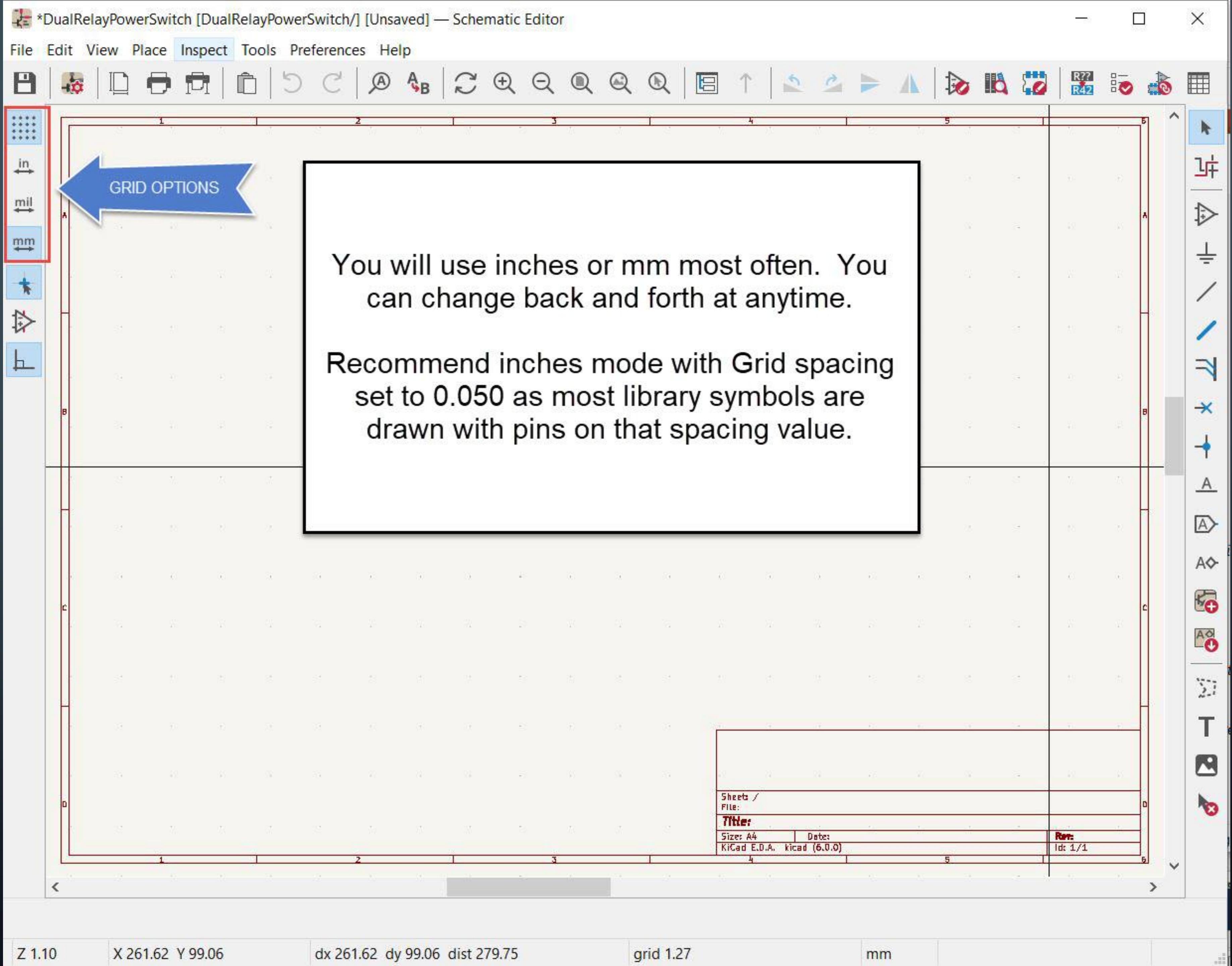
DRAW THE SCHEMATIC IN KICAD

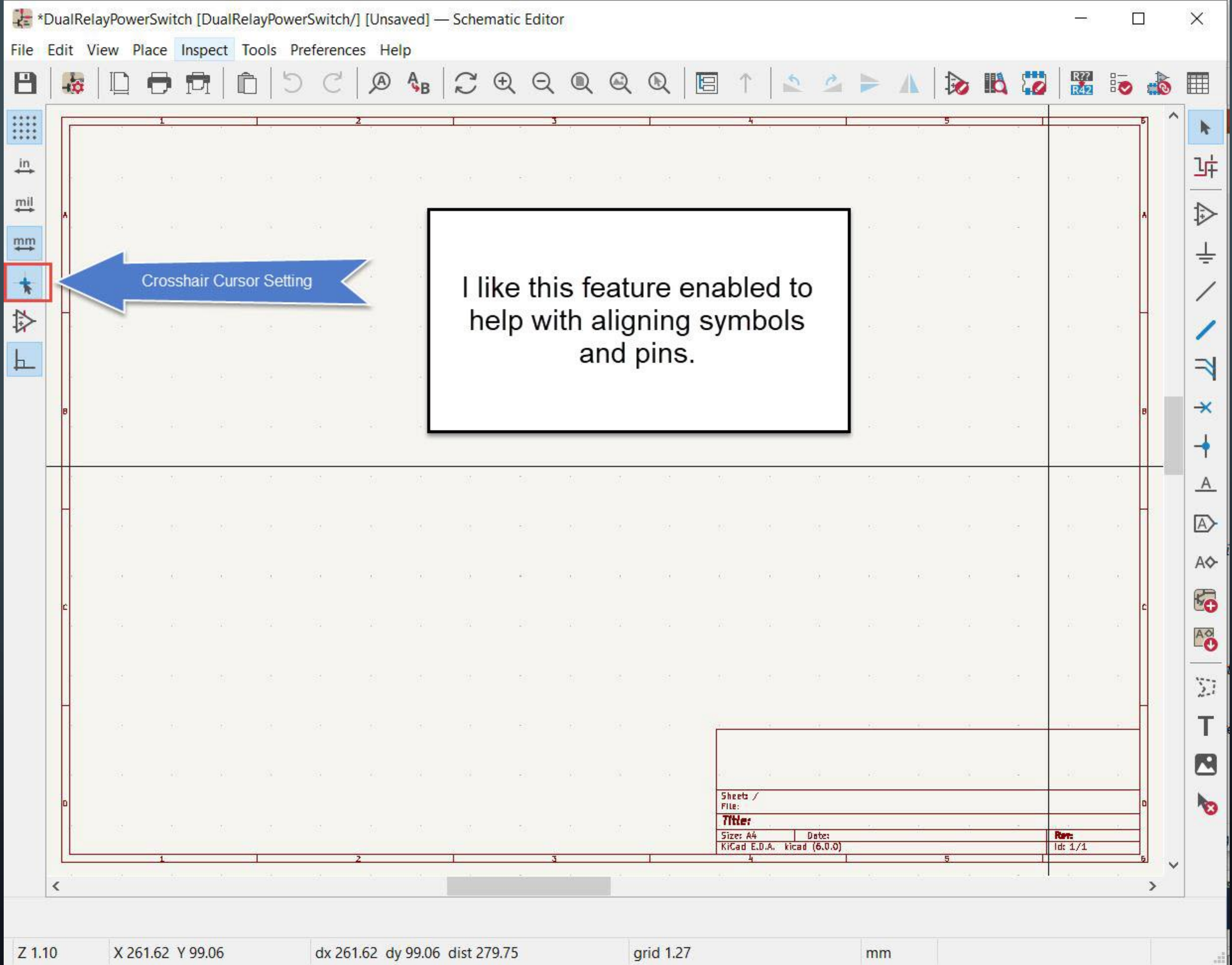
- We need to define the circuit in KiCad.   We do this by drawing the schematic in the Schematic Editor.
- This defines all of the connections between each component. We need that before designing the actual Printed Circuit Board (PCB)
- If not for anything else, you can use KiCad to draw nice, pretty schematics.
- In the KiCad Launcher, click on the Schematic Editor Icon.

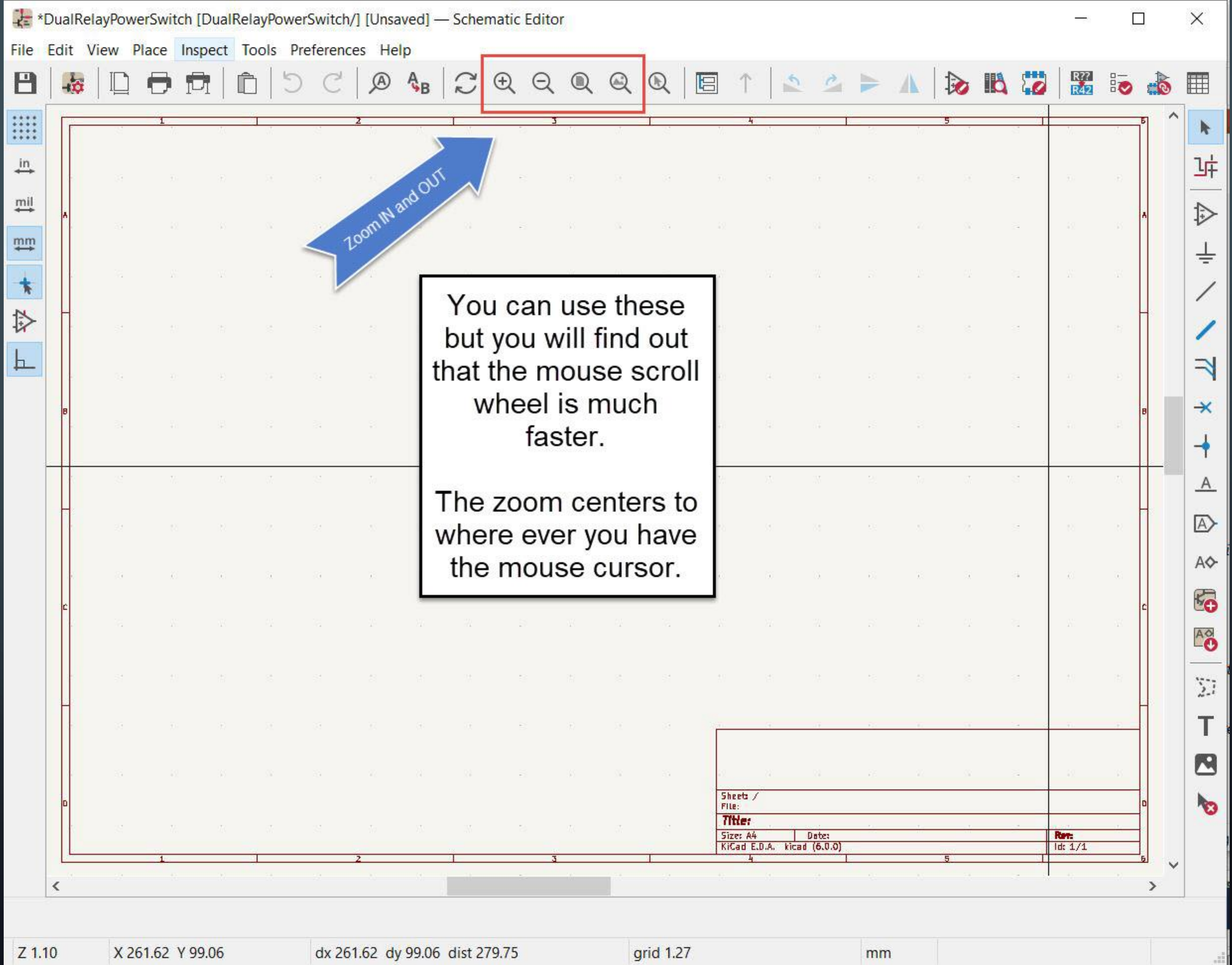


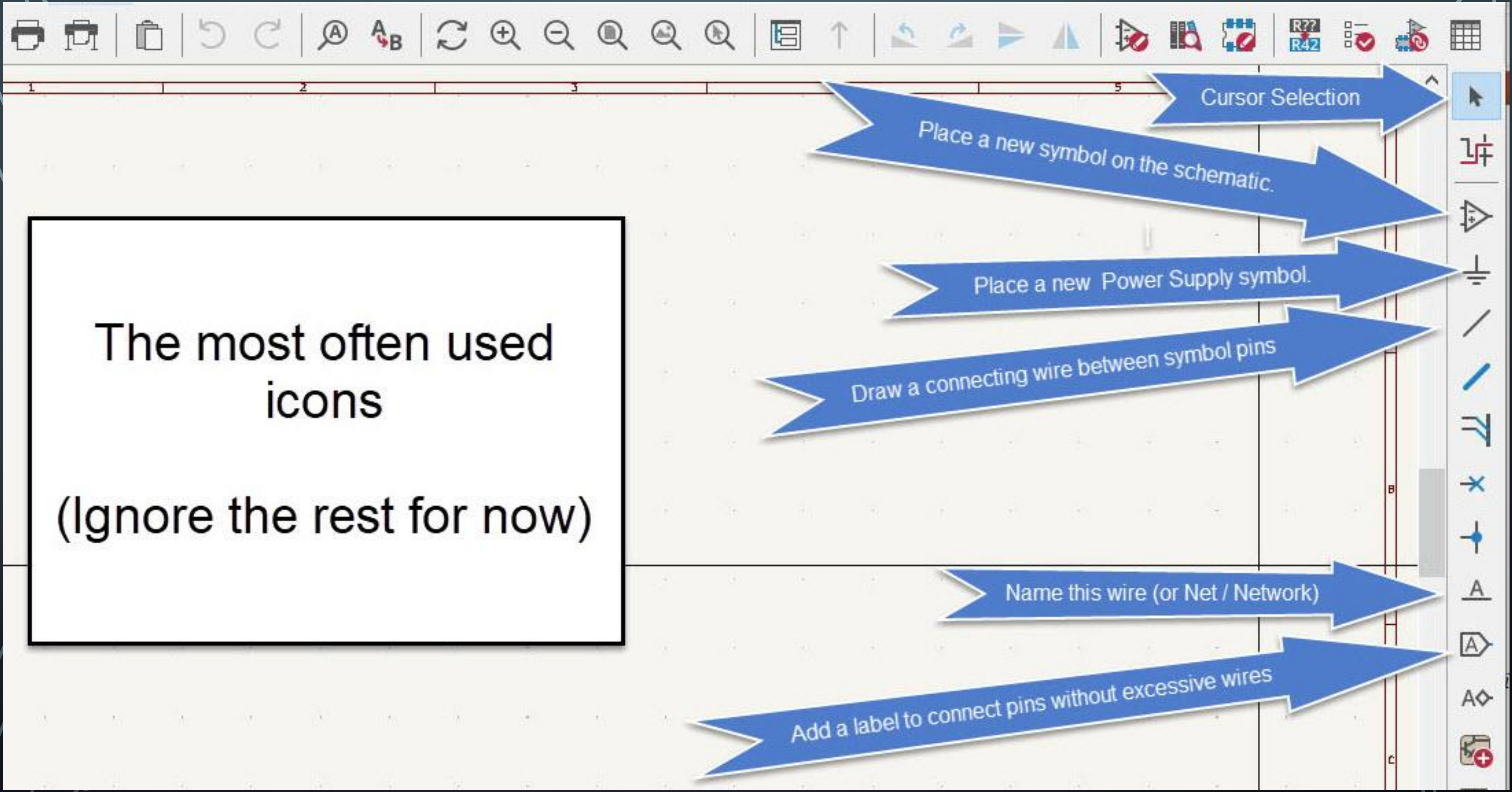
KICAD SCHEMATIC EDITOR

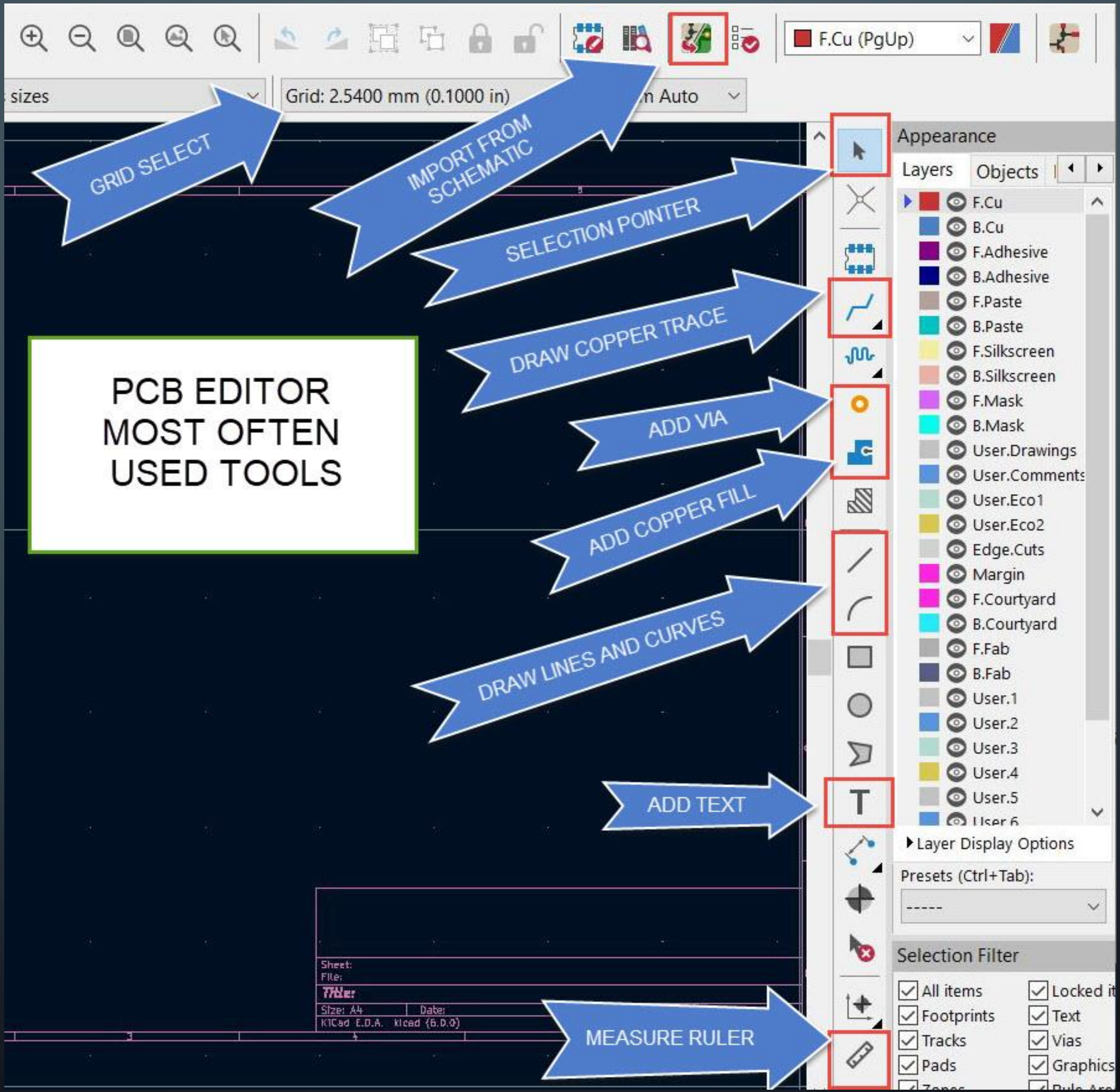


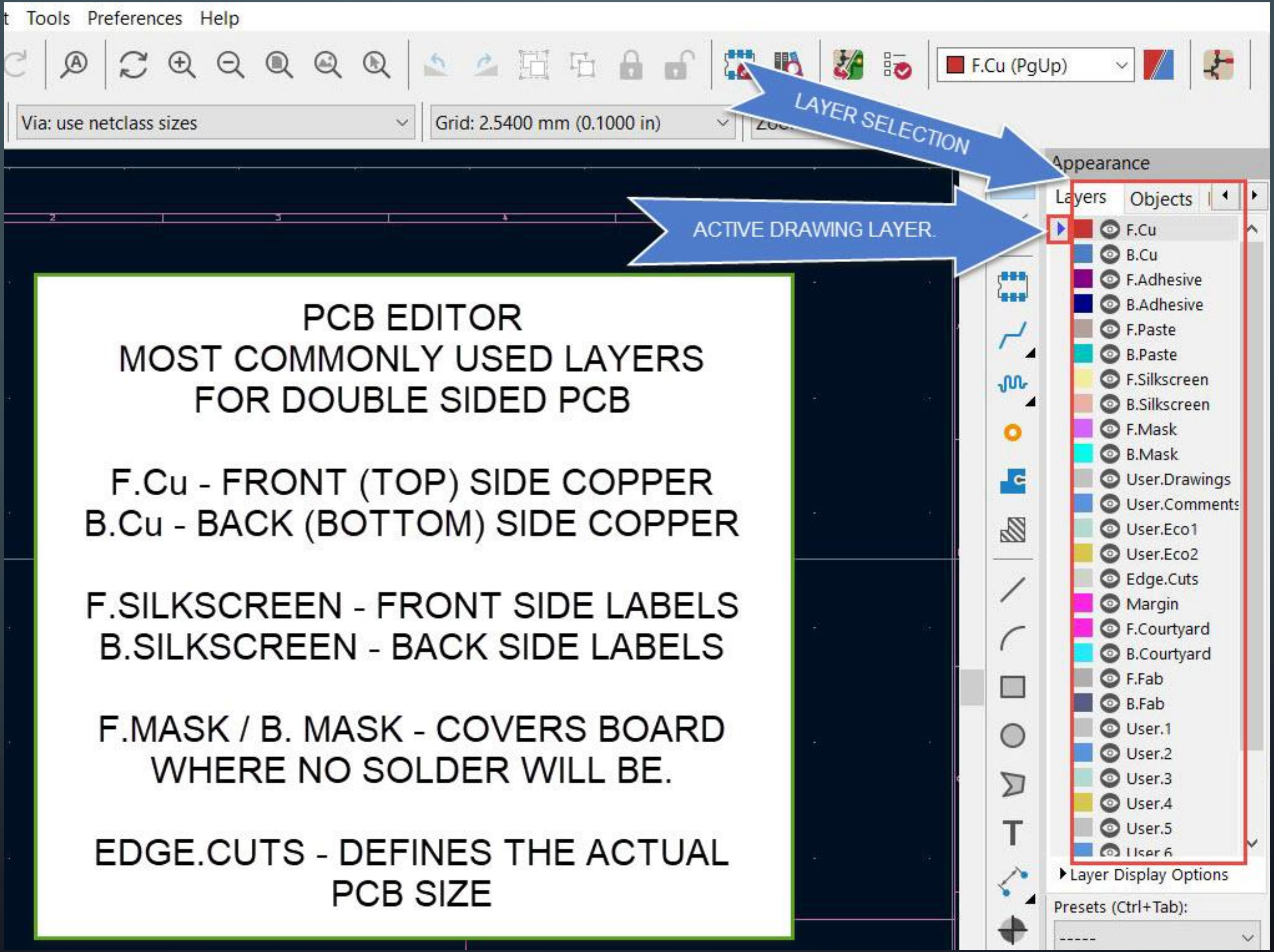

*DualRelayPowerSwitch [DualRelayPowerSwitch/] [Unsaved] — Schematic Editor
File Edit View Place Inspect Tools Preferences Help
GRID OPTIONS
You will use inches or mm most often. You can change back and forth at anytime.
Recommend inches mode with Grid spacing set to 0.050 as most library symbols are drawn with pins on that spacing value.
Z 1.10
X 261.62 Y 99.06
dx 261.62 dy 99.06 dist 279.75
grid 1.27
mm


*DualRelayPowerSwitch [DualRelayPowerSwitch/] [Unsaved] — Schematic Editor
File Edit View Place Inspect Tools Preferences Help
Crosshair Cursor Setting
I like this feature enabled to help with aligning symbols and pins.
Z 1.10
X 261.62 Y 99.06
dx 261.62 dy 99.06 dist 279.75
grid 1.27
mm


*DualRelayPowerSwitch [DualRelayPowerSwitch/] [Unsaved] — Schematic Editor
File Edit View Place Inspect Tools Preferences Help
Zoom IN and OUT
You can use these but you will find out that the mouse scroll wheel is much faster.
The zoom centers to where ever you have the mouse cursor.
Sheet: /
File:
Title:
Size: A4
Date:
KiCad E.D.A. kicad (6.0.0)
Rev:
Id: 1/1
Z 1.10
X 261.62 Y 99.06
dx 261.62 dy 99.06 dist 279.75
grid 1.27
mm


Cursor Selection
Place a new symbol on the schematic.
Place a new Power Supply symbol.
Draw a connecting wire between symbol pins
Name this wire (or Net / Network)
Add a label to connect pins without excessive wires
The most often used icons
(Ignore the rest for now)


PCB EDITOR MOST OFTEN USED TOOLS
GRID SELECT
IMPORT FROM SCHEMATIC
SELECTION POINTER
DRAW COPPER TRACE
ADD VIA
ADD COPPER FILL
DRAW LINES AND CURVES
ADD TEXT
MEASURE RULER
F.Cu (PgUp)
Grid: 2.5400 mm (0.1000 in)
Appearance
Layers
Objects
F.Cu
B.Cu
F.Adhesive
B.Adhesive
F.Paste
B.Paste
F.Silkscreen
B.Silkscreen
F.Mask
B.Mask
User.Drawings
User.Comments
User.Eco1
User.Eco2
Edge.Cuts
Margin
F.Courtyard
B.Courtyard
F.Fab
B.Fab
User.1
User.2
User.3
User.4
User.5
Layer Display Options
Presets (Ctrl+Tab):
Selection Filter
All items
Footprints
Tracks
Pads
Text
Vias
Graphics


Tools Preferences Help
F.Cu (PgUp)
Via: use netclass sizes
Grid: 2.5400 mm (0.1000 in)
LAYER SELECTION
ACTIVE DRAWING LAYER.
Appearance
Layers Objects
F.Cu
B.Cu
F.Adhesive
B.Adhesive
F.Paste
B.Paste
F.Silkscreen
B.Silkscreen
F.Mask
B.Mask
User.Drawings
User.Comments
User.Eco1
User.Eco2
Edge.Cuts
Margin
F.Courtyard
B.Courtyard
F.Fab
B.Fab
User.1
User.2
User.3
User.4
User.5
Layer Display Options
Presets (Ctrl+Tab):

PCB EDITOR
MOST COMMONLY USED LAYERS
FOR DOUBLE SIDED PCB
F.Cu - FRONT (TOP) SIDE COPPER
B.Cu - BACK (BOTTOM) SIDE COPPER
F.SILKSCREEN - FRONT SIDE LABELS
B.SILKSCREEN - BACK SIDE LABELS
F.MASK / B. MASK - COVERS BOARD
WHERE NO SOLDER WILL BE.
EDGE.CUTS - DEFINES THE ACTUAL
PCB SIZE

NOW, LET'S GO INTO KICAD AND POKE AROUND

RESOURCES

- KiCad website https://www.kicad.org/
- Printed Circuit Board – https://en.wikipedia.org/wiki/Printed_circuit_board
- Digi-Key KiCad library https://www.digikey.com/en/resources/design-tools/kicad

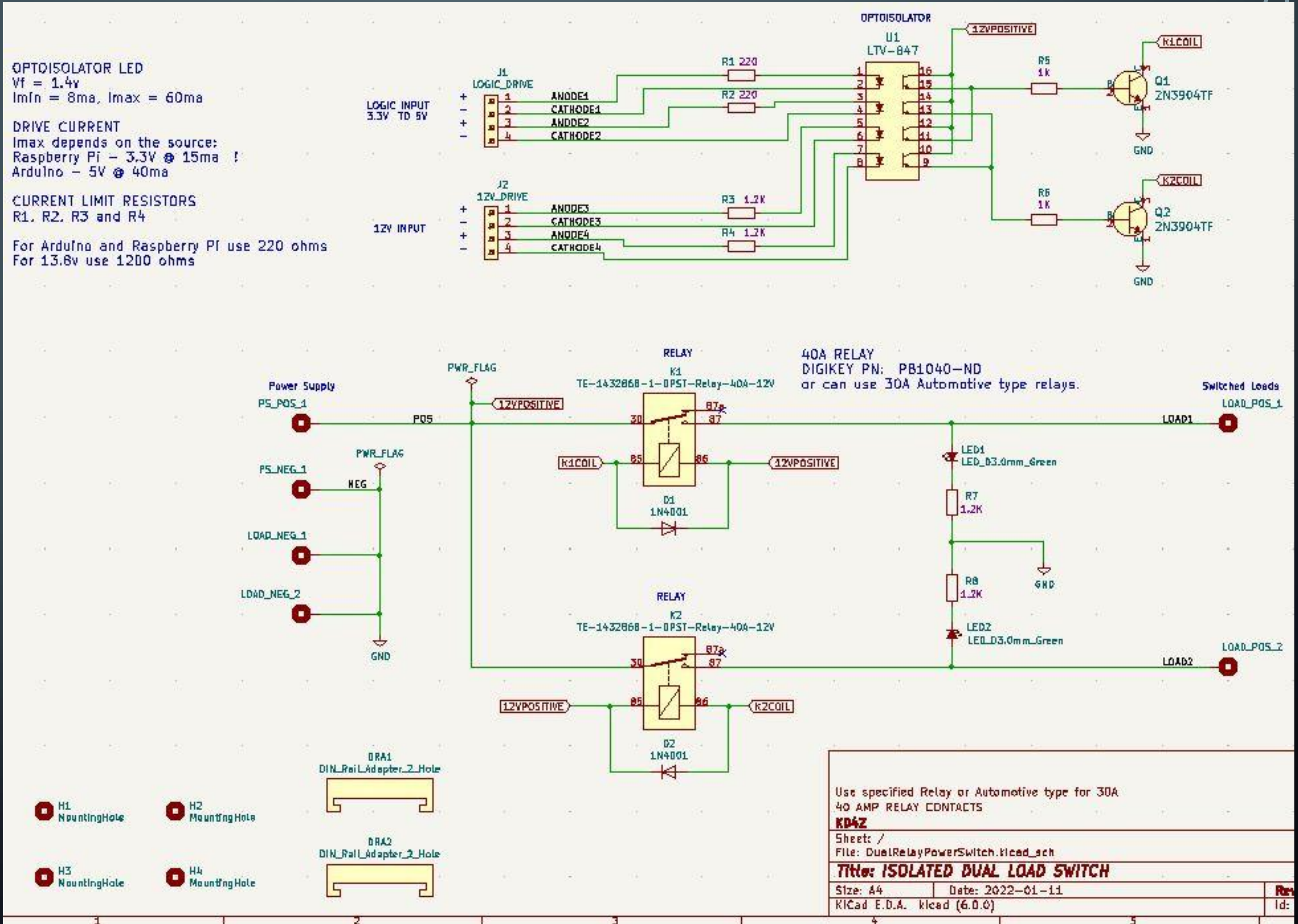
Fabrication:

- JLCPCB https://jlcpcb.com/ PCBWay https://www.pcbway.com/
- Oshpark https://oshpark.com/

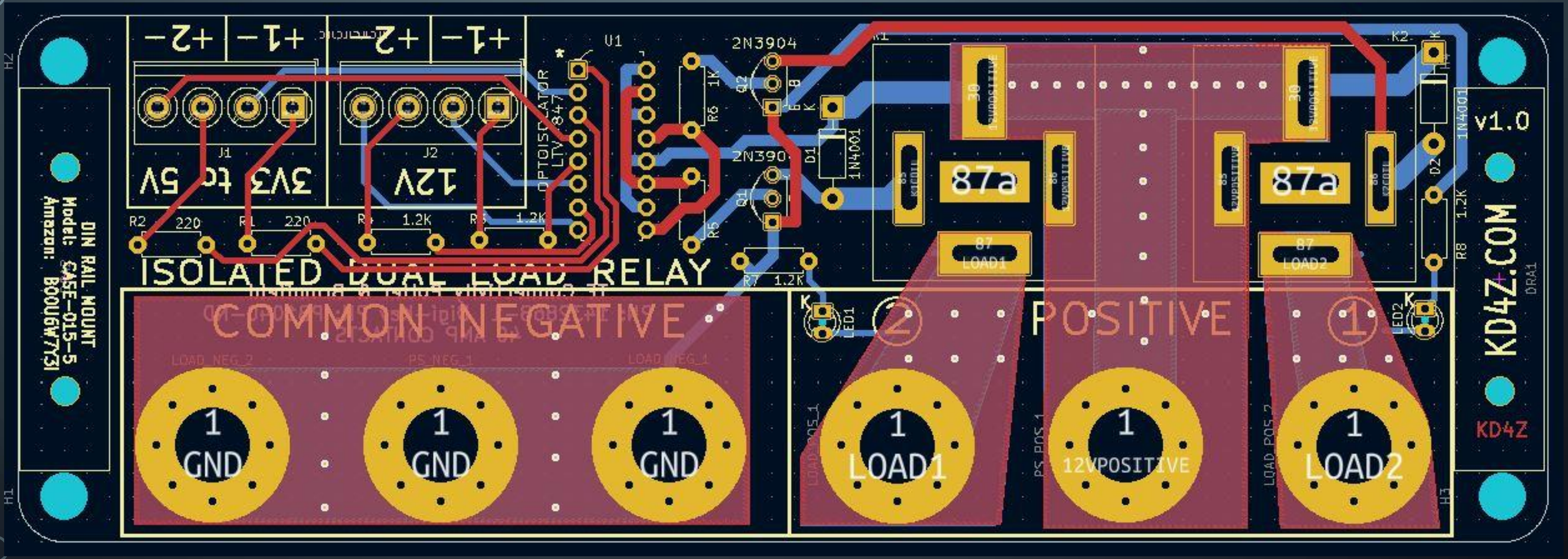
- KiCad Presentation available on my QRZ page https://www.qrz.com/db/kd4z

FINAL SCHEMATIC

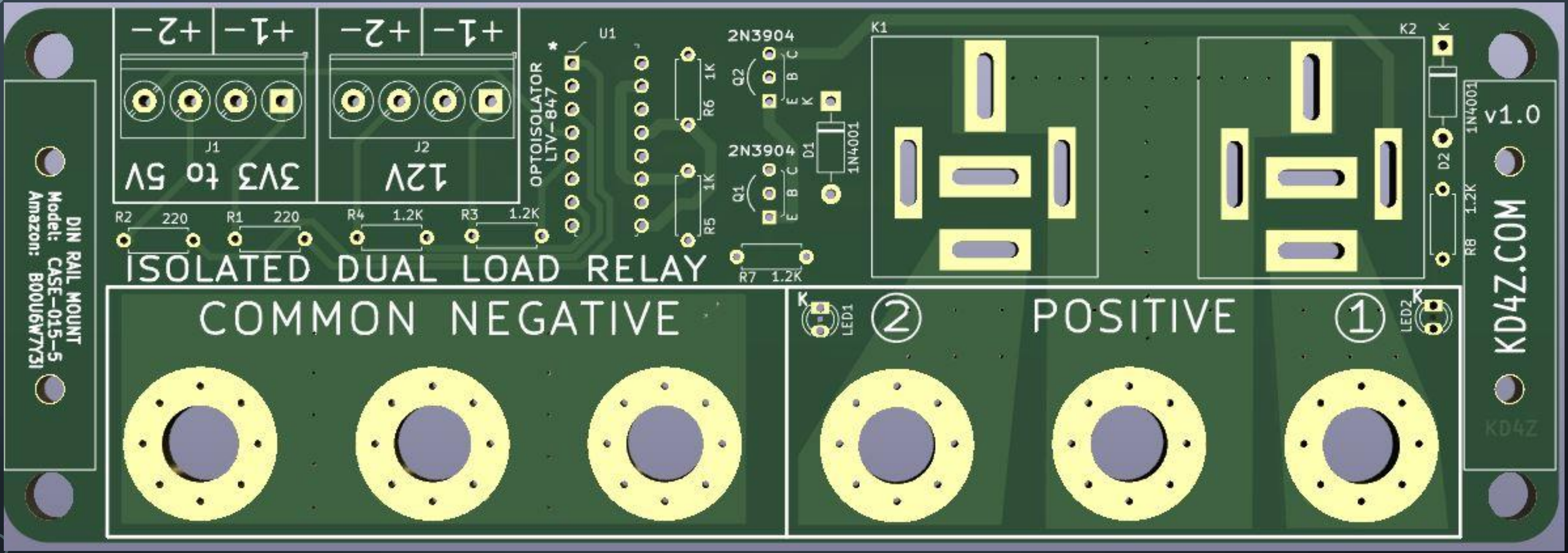
KD4Z - KICAD JUMPSTART 1 - 16 FEB 2022



FINAL PCB LAYOUT



FINAL PCB LAYOUT 3D – TOP SIDE



FINAL PCB LAYOUT 3D – BOTTOM SIDE