“KiCAD for Robots” – A Tutorial

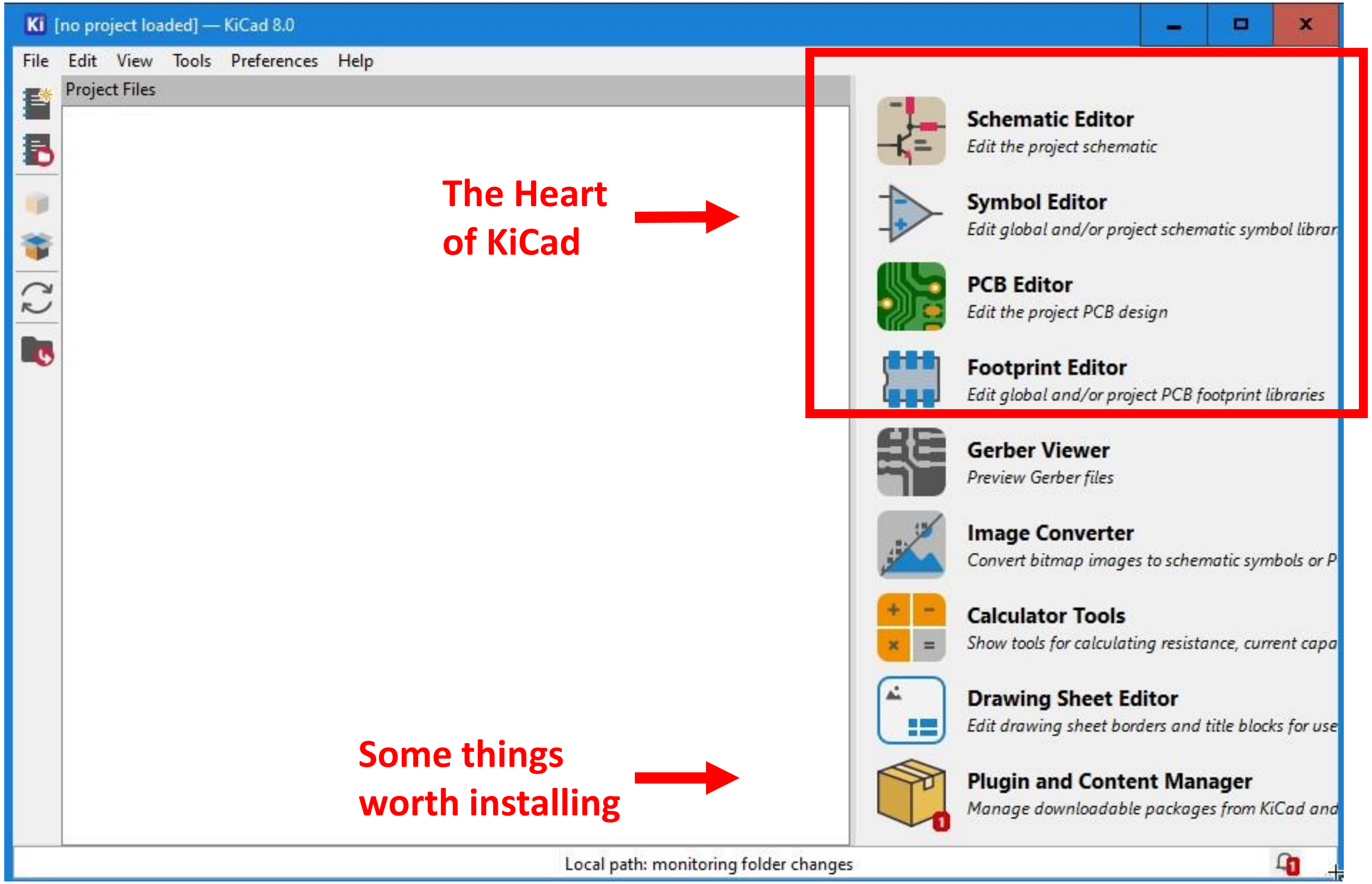
Presented by
Doug Paradis

Download KiCad: https://www.kicad.org/download/

What are we going to cover?

- There are four parts to the presentation:
  - Intro to KiCad’s interface
  - How to build a schematic
  - How to layout a PCB
  - How to get a PCB built at JLCPCB
- Our example PCB will be a “Module Motherboard”.
- Small cheap easy to use electronic modules exist for most functions used on a robot.
- Wiring many modules together without a PCB can turn into an unreliable “rat’s nest” of wires, and is prone to mistakes.
- The techniques learned can be used on any PCB.

KiCad Opening Screen



Schematic Editor
Edit the project schematic

Symbol Editor
Edit global and/or project schematic symbol librar

PCB Editor
Edit the project PCB design

Footprint Editor
Edit global and/or project PCB footprint libraries

Gerber Viewer
Preview Gerber files

Image Converter
Convert bitmap images to schematic symbols or P

Calculator Tools
Show tools for calculating resistance, current capa

Drawing Sheet Editor
Edit drawing sheet borders and title blocks for use

Plugin and Content Manager
Manage downloadable packages from KiCad and

Local path: monitoring folder changes

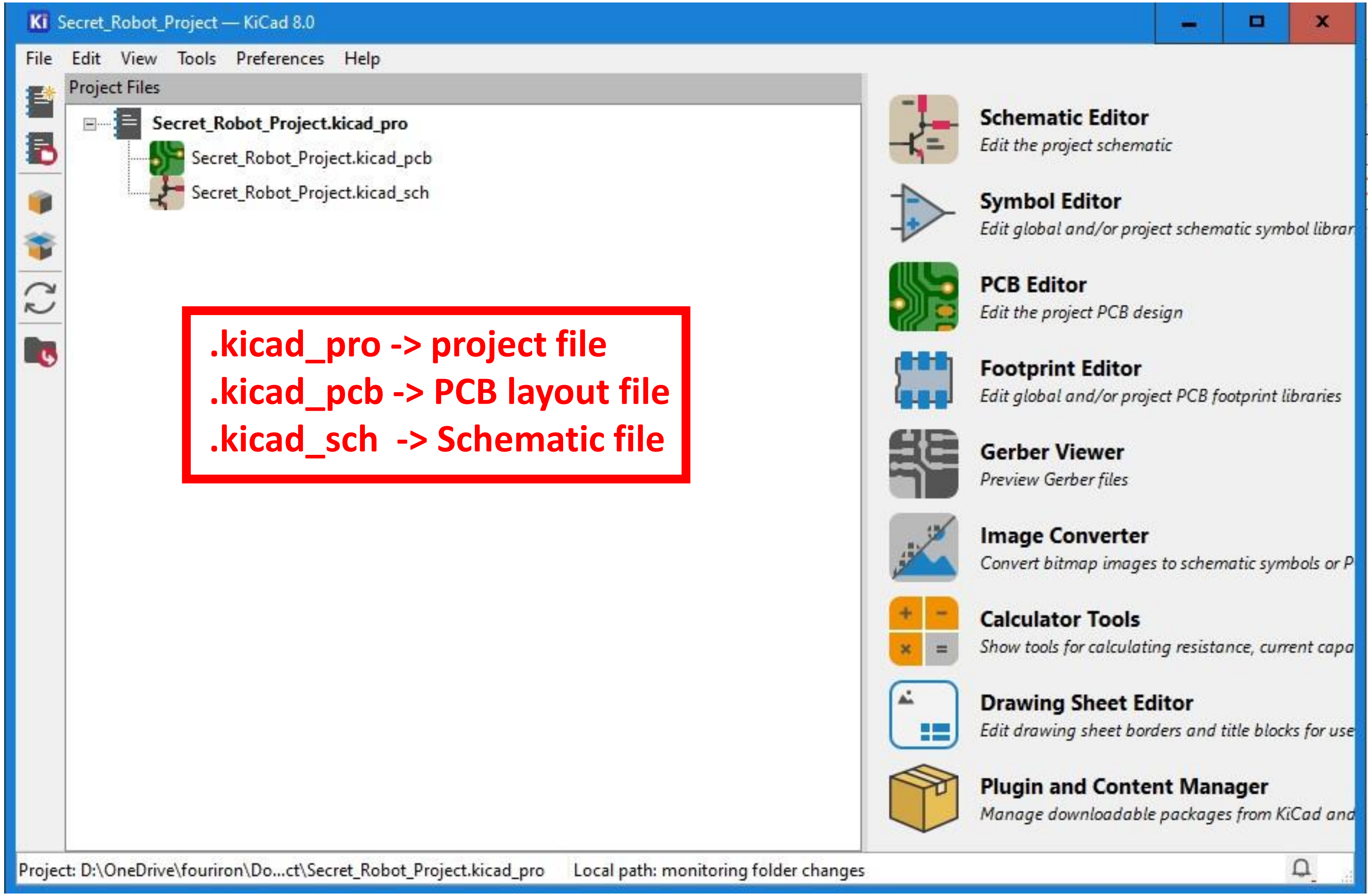
The Heart of KiCad

Some things worth installing

1. Select the "Plugin and Content Manager".
2. Install the "Freerouting" and "Git" plugins.

New Project

- Create a “KiCad_Projects” directory and a sub-directory for your project.
- In the project directory create 4 sub-directories
  - Project
  - Symbols
  - Footprints
  - 3D-models
- In KiCad, File>New Project.
- Steer to your directory tree and enter project name in the Project directory. We are using “Secret_Robot_Project”.
- Hit Enter and you should see the screen to the Right.



Secret_Robot_Project — KiCad 8.0
File
Edit
View
Tools
Preferences
Help
Project Files
Schematic Editor
Edit the project schematic
Symbol Editor
Edit global and/or project schematic symbol librar
PCB Editor
Edit the project PCB design
Footprint Editor
Edit global and/or project PCB footprint libraries
Gerber Viewer
Preview Gerber files
Image Converter
Convert bitmap images to schematic symbols or P
Calculator Tools
Show tools for calculating resistance, current capa
Drawing Sheet Editor
Edit drawing sheet borders and title blocks for use
Plugin and Content Manager
Manage downloadable packages from KiCad and

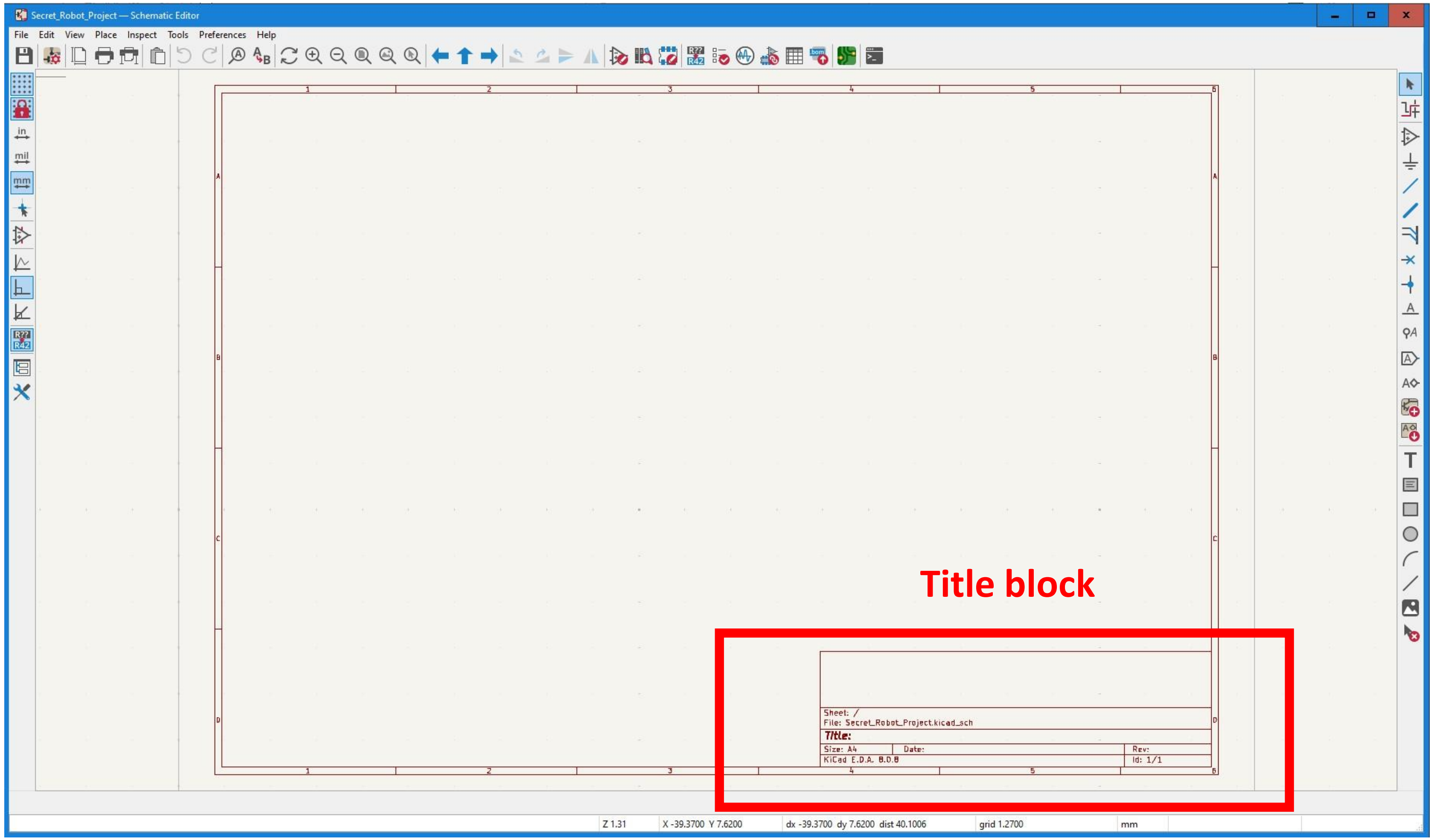
Secret_Robot_Project.kicad_pro
- Secret_Robot_Project.kicad_pcb
- Secret_Robot_Project.kicad_sch

Project: D:\OneDrive\fouriron\Do...ct\Secret_Robot_Project.kicad_pro   Local path: monitoring folder changes

.kicad_pro -> project file
.kicad_pcb -> PCB layout file
.kicad_sch -> Schematic file

New Project

- **Click on the Schematic Editor icon or the .kicad_sch file icon.**
- **The screen to the right appears.**
- **This is the Schematic Screen.**
- **The center is the schematic sheet (like it was in paper times…).**
- **The lower right corner has the sheet ‘s title block.**



Secret_Robot_Project — Schematic Editor

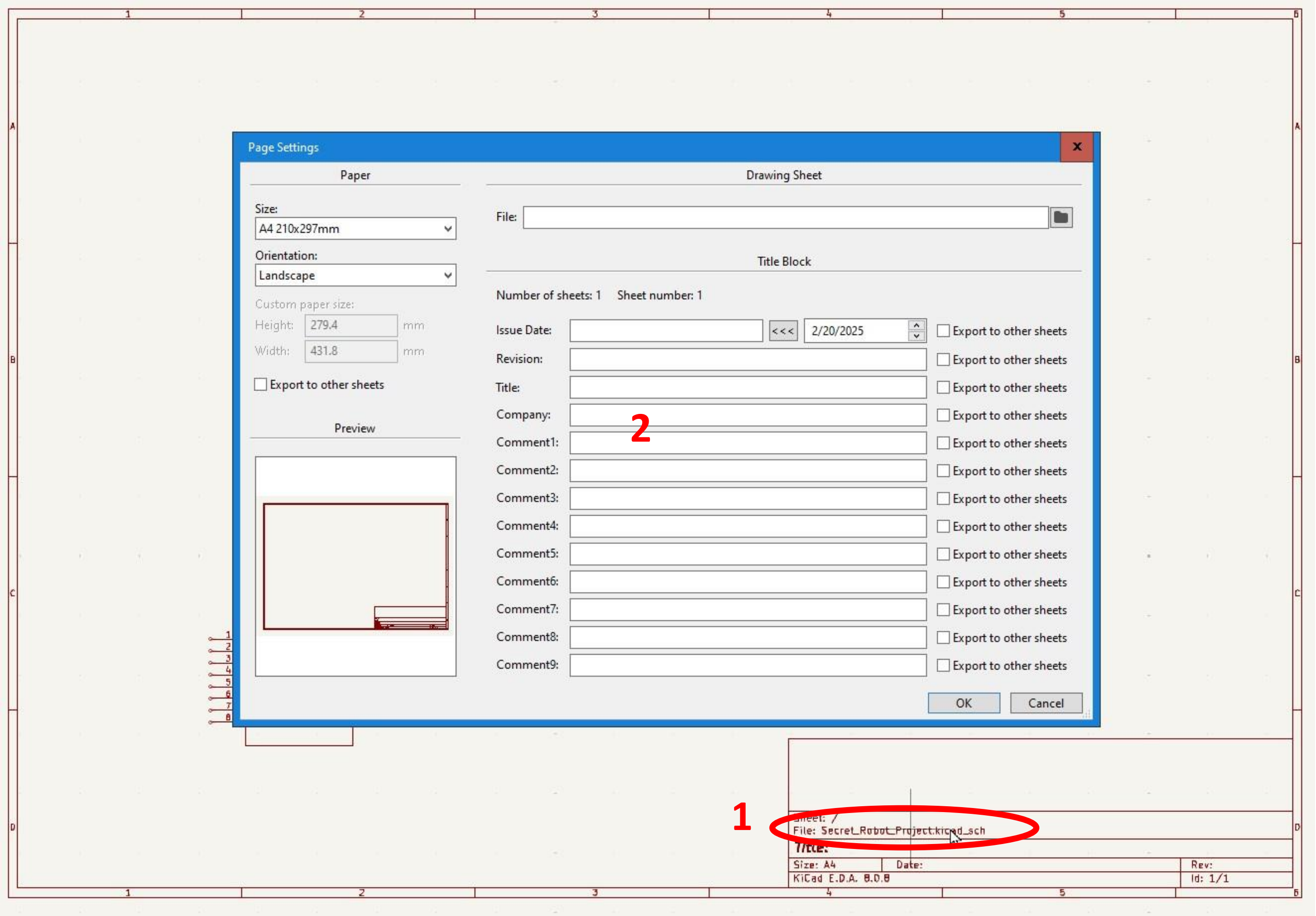
Title block

Sheet: /
File: Secret_Robot_Project.kicad_sch
Title:
Size: A4 Date: Rev:
KiCad E.D.A. 8.0.8 Id: 1/1

Z 1.31 X -39.3700 Y 7.6200 dx -39.3700 dy 7.6200 dist 40.1006 grid 1.2700 mm

Editing the Title Block

1. Click on the file name field in the Title Block.
2. In the Page Setting window edit the fields, as you desire.
3. You can also use File>Page Settings.
4. You should change the page size from A3 to US letter.



Page Settings

Paper

Size: A4 210x297mm

Orientation: Landscape

Custom paper size:

Height: 279.4 mm

Width: 431.8 mm

□ Export to other sheets

Preview

Drawing Sheet

File:

Title Block

Number of sheets: 1 Sheet number: 1

Issue Date: <<< 2/20/2025 □ Export to other sheets

Revision: □ Export to other sheets

Title: □ Export to other sheets

Company: □ Export to other sheets

Comment1: □ Export to other sheets

Comment2: □ Export to other sheets

Comment3: □ Export to other sheets

Comment4: □ Export to other sheets

Comment5: □ Export to other sheets

Comment6: □ Export to other sheets

Comment7: □ Export to other sheets

Comment8: □ Export to other sheets

Comment9: □ Export to other sheets

OK Cancel

Sheet: /

File: Secret_Robot_Project.kicad_sch

Title:

Size: A4 Date: Rev:

KiCad E.D.A. 8.0.8 Id: 1/1

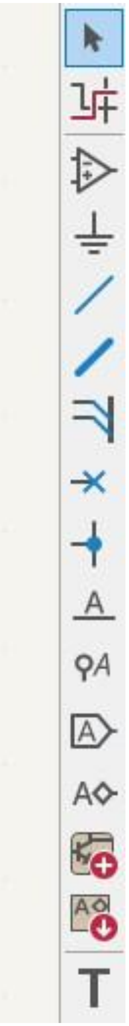
New Project

- The right menu has what you need to make a schematic.

- About halfway down you have the “labels”. They vary in scope depending on the type.
  - Global labels connect globally.
  - Net labels only connect on the same sheet.
  - Hierarchical labels connect within sheet borders and to hierarchical sheet boxes.

- In this session, we will not cover hierarchical sheets, but to show a schematic made with them.

- Below the shown icons are some icons that deal with drawing shapes on the schematic sheet.



<- select item - IMPORTANT

<- highlight nets

<- add symbol - IMPORTANT

<- add Power sym - IMPORTANT

<- add wire - IMPORTANT

<- add bus

<- add wire to bus

<- add “no connect” flg - IMPORTANT

<- add junction

<- add net label - IMPORTANT

<- add net class directive-IMPORTANT

<- add global label - IMPORTANT

<- add hierarchical label – important but…

<- add sheet (hierarchical) – important but…

<- add sheet pin (hierarchical) –

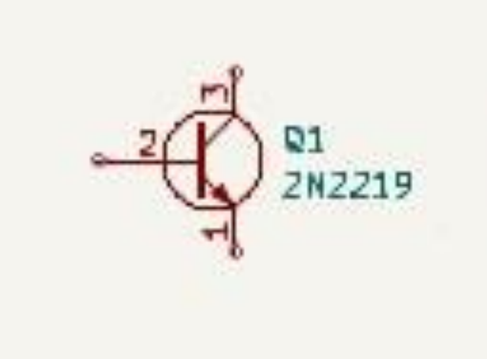
<- add text – nice for adding comments

What is a “Symbol”

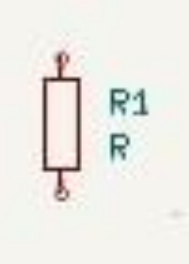
A symbol is a diagram that is a representation of the part and its pinout. It is a schematic “symbol”.

Examples:

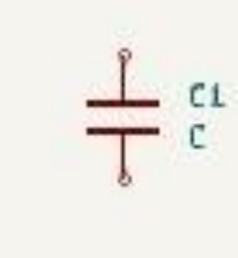
Transistor (NPN)



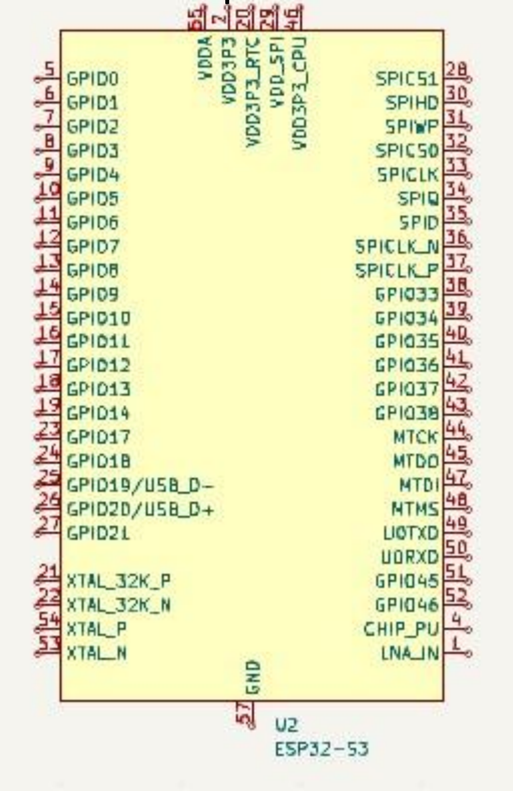
Resistor



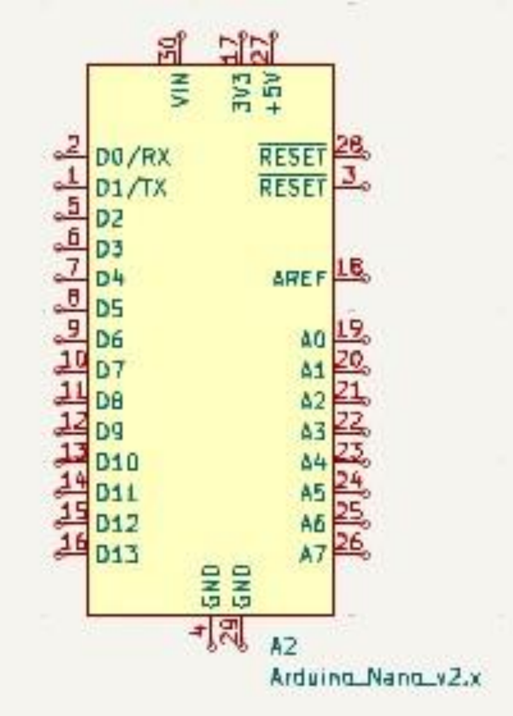
Capacitor



Microprocessor



Module

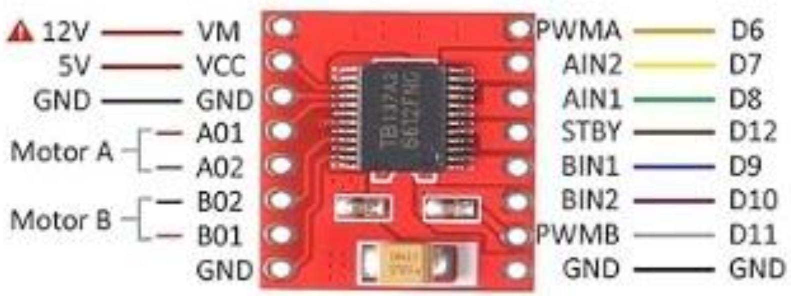


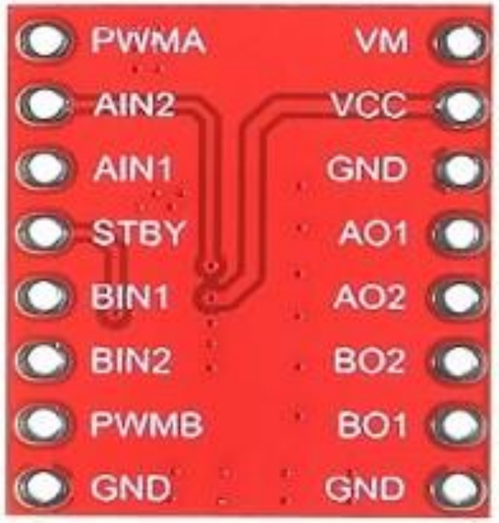
Custom Module

Either you make your own or you find the module symbol on the internet.

Custom Symbols

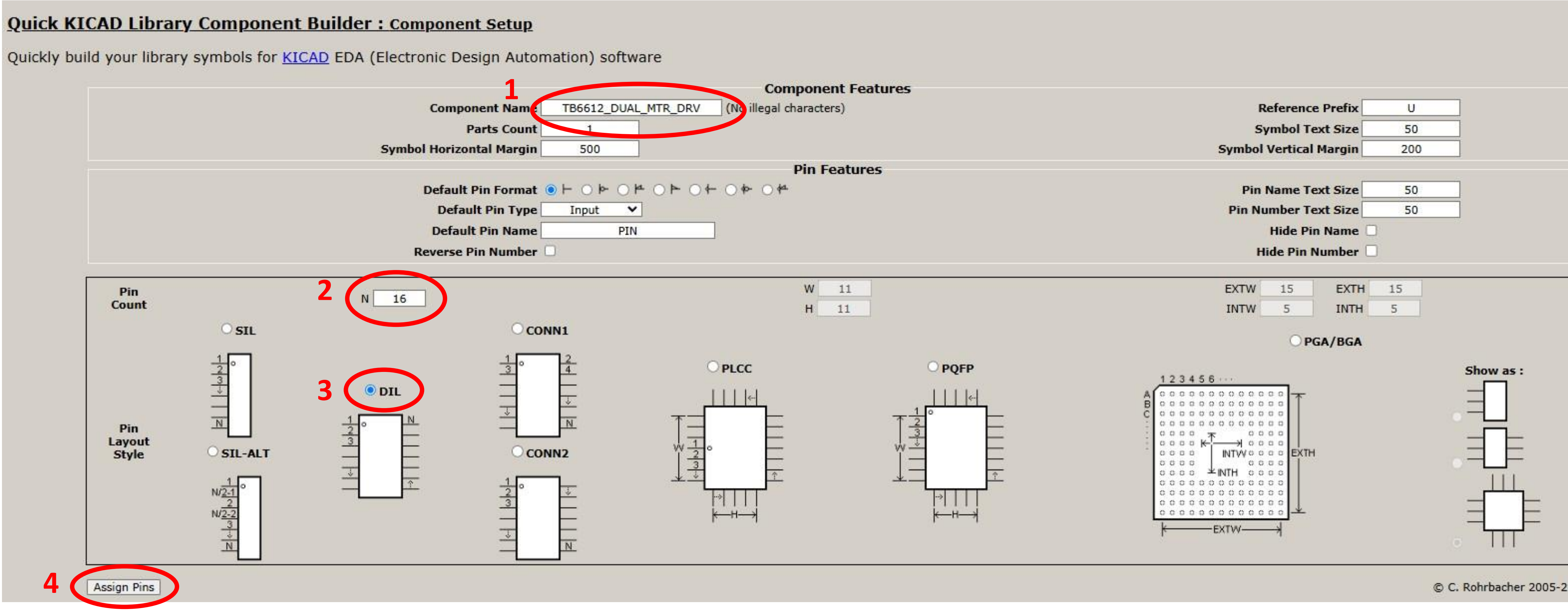
- **We want to use a TB6612FNA Dual Motor Driver. It doesn’t have a symbol in KiCad. We need to make one.**





KiCAD Library Component Builder

http://kicad.rohrbacher.net/quicklib.php



Quick KICAD Library Component Builder : Component Setup

Quickly build your library symbols for KICAD EDA (Electronic Design Automation) software

Component Features

1 Component Name	TB6612_DUAL_MTR_DRV (No illegal characters)	Reference Prefix	U
Parts Count	1	Symbol Text Size	50
Symbol Horizontal Margin	500	Symbol Vertical Margin	200

Pin Features

Default Pin Format	● ○ ○ ○ ○ ○ ○	Pin Name Text Size	50
Default Pin Type	Input	Pin Number Text Size	50
Default Pin Name	PIN	Hide Pin Name	□
Reverse Pin Number	□	Hide Pin Number	□

Pin Count: 2 N 16

W 11 H 11 EXTW 15 EXTH 15 INTW 5 INTH 5

Pin Layout Style: ○ SIL ○ SIL-ALT 3 ● DIL ○ CONN1 ○ CONN2 ○ PLCC ○ PQFP ○ PGA/BGA

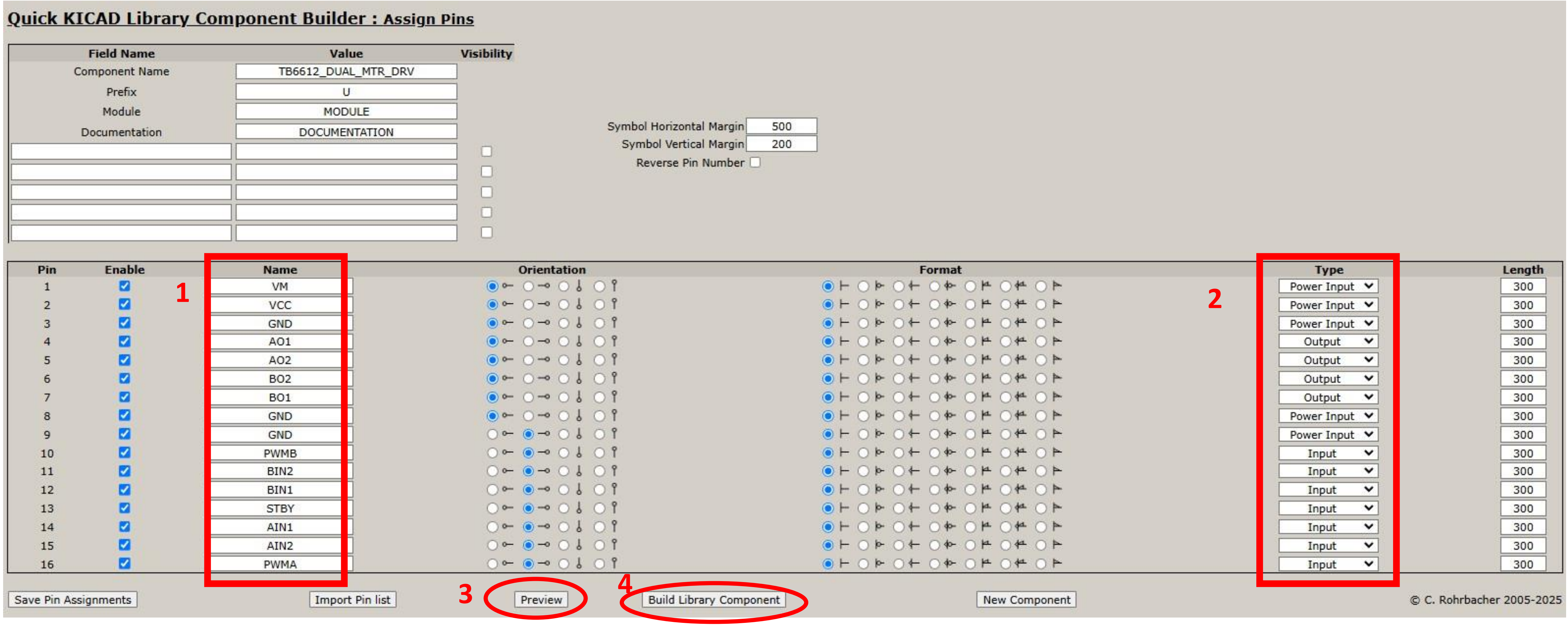
Show as :

4 Assign Pins

© C. Rohrbacher 2005-2

The other settings can usually use the default values.

KiCAD Library Component Builder



Quick KICAD Library Component Builder : Assign Pins

Field Name	Value	Visibility
Component Name	TB6612_DUAL_MTR_DRV	
Prefix	U	
Module	MODULE	
Documentation	DOCUMENTATION	
		☐
		☐
		☐
		☐
		☐

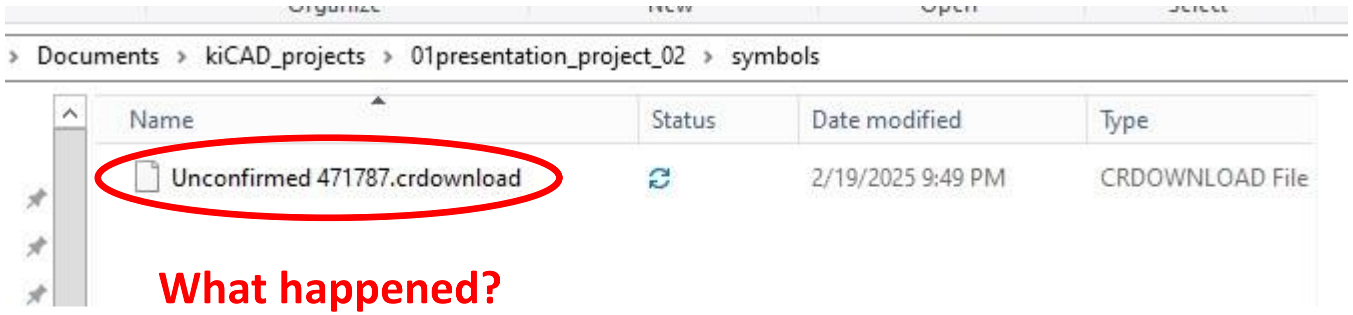
Symbol Horizontal Margin 500
Symbol Vertical Margin 200
Reverse Pin Number ☐

Pin	Enable	Name	Orientation	Format	Type	Length
1	☑	VM	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Power Input	300
2	☑	VCC	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Power Input	300
3	☑	GND	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Power Input	300
4	☑	AO1	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Output	300
5	☑	AO2	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Output	300
6	☑	BO2	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Output	300
7	☑	BO1	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Output	300
8	☑	GND	● ○ ○ ○	● ○ ○ ○ ○ ○ ○	Power Input	300
9	☑	GND	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Power Input	300
10	☑	PWMB	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Input	300
11	☑	BIN2	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Input	300
12	☑	BIN1	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Input	300
13	☑	STBY	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Input	300
14	☑	AIN1	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Input	300
15	☑	AIN2	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Input	300
16	☑	PWMA	○ ● ○ ○	● ○ ○ ○ ○ ○ ○	Input	300

© C. Rohrbacher 2005-2025

When you download the symbol, make sure to click on "keep".

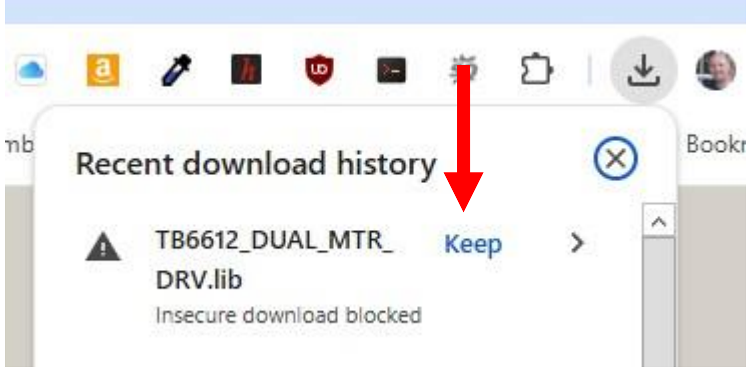
Saving the Symbol We Made



Documents › kiCAD_projects › 01presentation_project_02 › symbols

Name	Status	Date modified	Type
Unconfirmed 471787.crdownload		2/19/2025 9:49 PM	CRDOWNLOAD File

What happened?

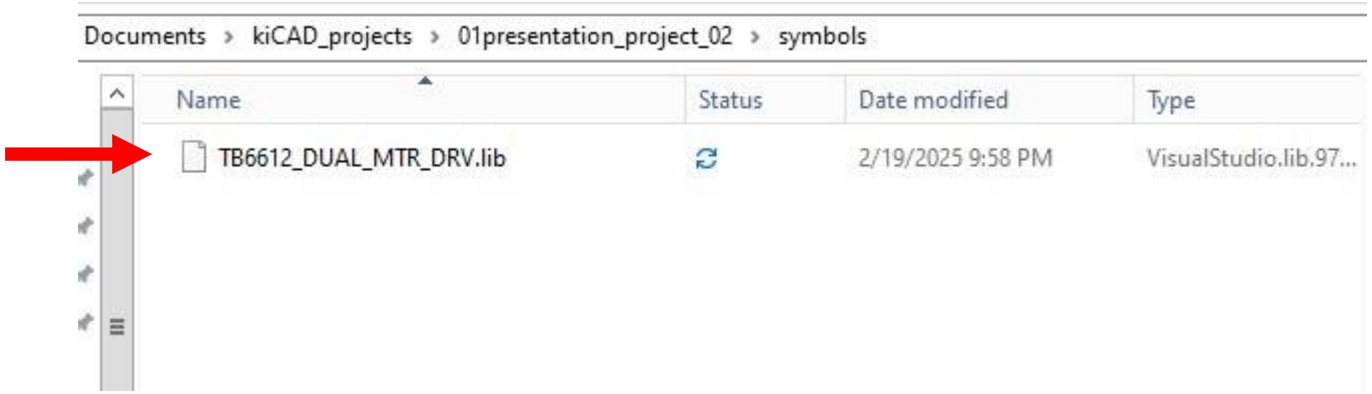


Recent download history

TB6612_DUAL_MTR_DRV.lib

Insecure download blocked

Chrome will not automatically download a file form a non-https website. You have to click on the Keep.



Documents › kiCAD_projects › 01presentation_project_02 › symbols

Name	Status	Date modified	Type
TB6612_DUAL_MTR_DRV.lib		2/19/2025 9:58 PM	VisualStudio.lib.97...

Setting Symbol Pin Type

Pin Types:

- Input
- Output
- Bidir
- Tri-state
- Passive
- Unspecified
- Power Input
- Power Output
- Open Collector
- Open Emitter

What are electrical types used for?

The electrical type is used by the electrical rule check (ERC) to determine if you made an obvious mistake. ERC is high, limited. All it does is read all electrical types of pins connected to a particular net and check their combination against the error matrix.

Example: By high, connecting two output pins together will result in an error. (Makes sense as one output can be **high**, and one can be **low** resulting in a short circuit between the positive supply and ground.

Author: Rene Posch – Aug 2020

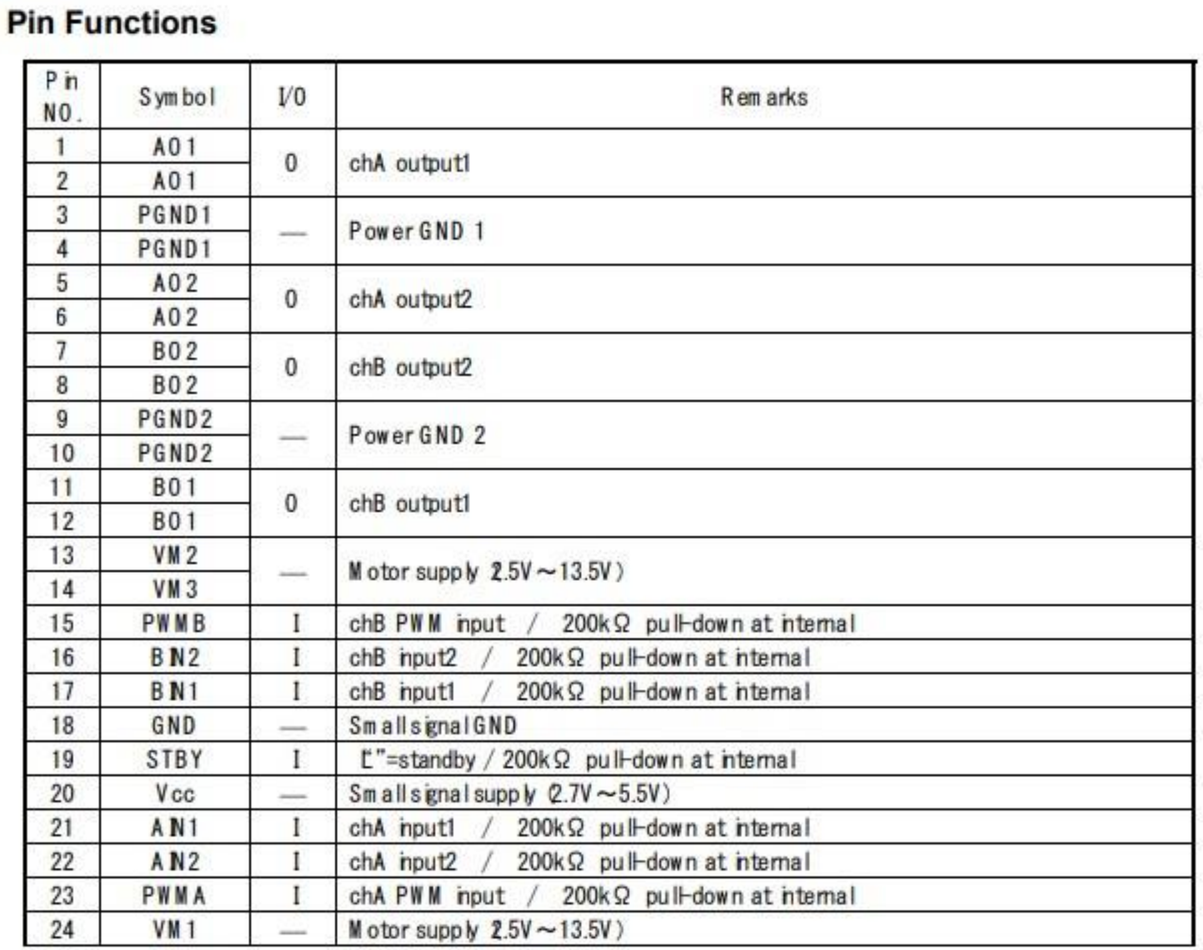
For some details - https://forum.kicad.info/t/electrical-type-of-schematic-symbol-pins-kicad-4-and-kicad-5/9439

Setting Symbol Pin Type

To make ERC as useful as possible one needs to take care which electrical types to assign to pins.

The datasheet of your component is a good start for this.

From the TB6612FNG datasheet



Pin Functions

Pin NO.	Symbol	I/O	Remarks
1	AO1	O	chA output1
2	AO1		
3	PGND1	—	Power GND 1
4	PGND1		
5	AO2	O	chA output2
6	AO2		
7	BO2	O	chB output2
8	BO2		
9	PGND2	—	Power GND 2
10	PGND2		
11	BO1	O	chB output1
12	BO1		
13	VM2	—	Motor supply (2.5V～13.5V)
14	VM3		
15	PWMB	I	chB PWM input / 200kΩ pull-down at internal
16	BIN2	I	chB input2 / 200kΩ pull-down at internal
17	BIN1	I	chB input1 / 200kΩ pull-down at internal
18	GND	—	Small signal GND
19	STBY	I	“L”=standby / 200kΩ pull-down at internal
20	Vcc	—	Small signal supply (2.7V～5.5V)
21	AIN1	I	chA input1 / 200kΩ pull-down at internal
22	AIN2	I	chA input2 / 200kΩ pull-down at internal
23	PWMA	I	chA PWM input / 200kΩ pull-down at internal
24	VM1	—	Motor supply (2.5V～13.5V)

Setting Symbol Pin Type

Author: Rene Posch – Aug 2020
https://forum.kicad.info/t/electrical-type-of-schematic-symbol-pins-kicad-4-and-kicad-5/9439

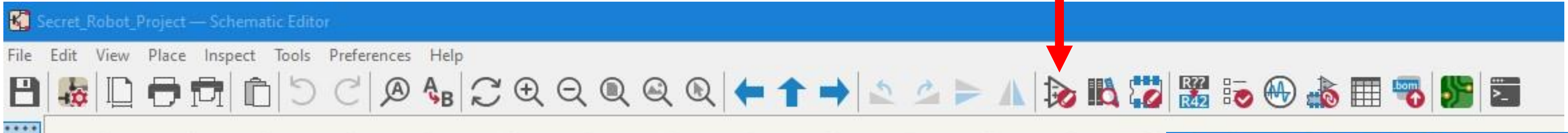
To make ERC as useful as possible one needs to take care which electrical types to assign to pins. The datasheet of your component is a good start for this.

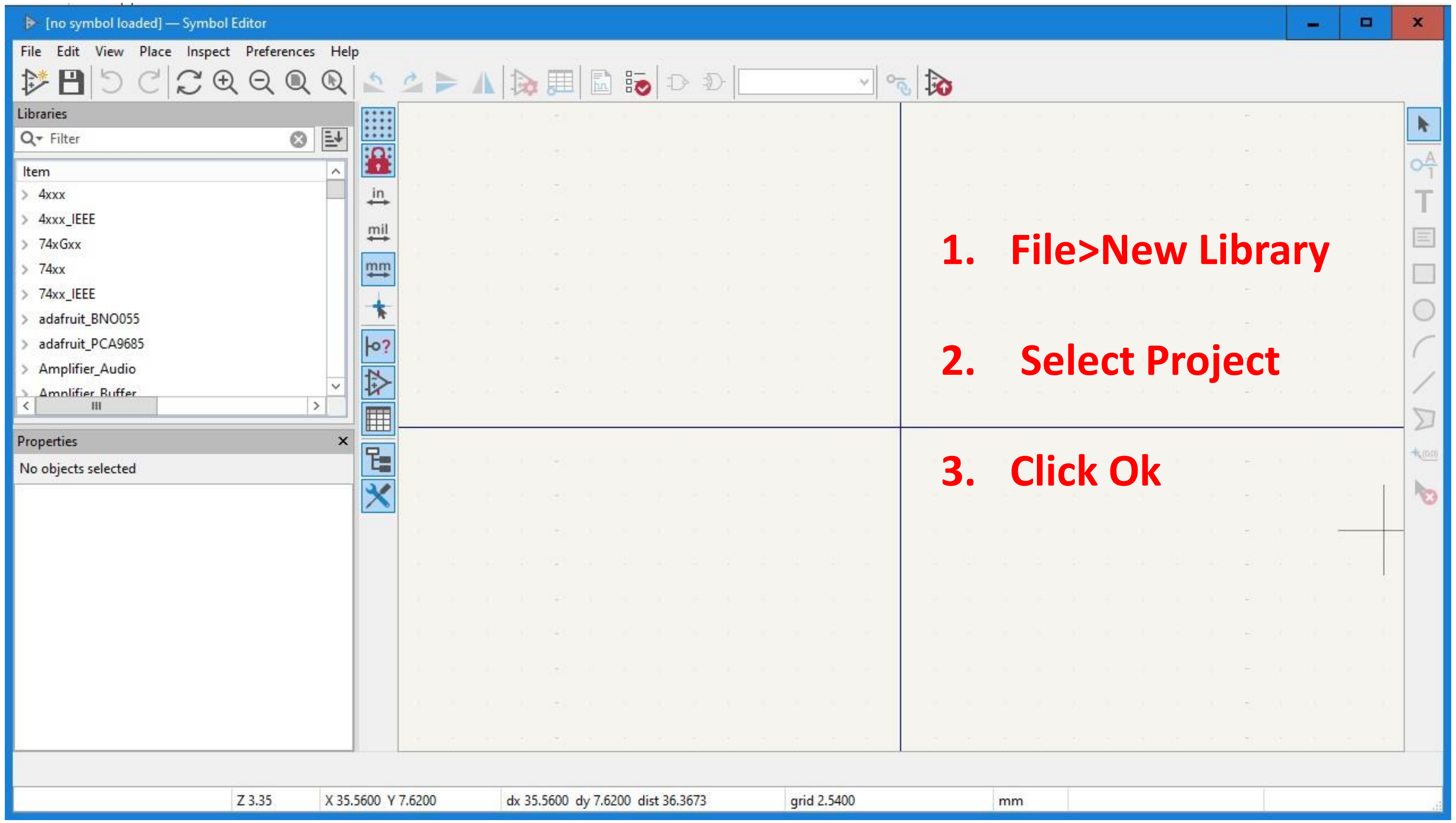
- Pins through which your component is supplied (vcc, gnd, vss, …) are power inputs.
- Digital and analog input pins are inputs.
- Digital and analog output pins are output (For digital pins, use this only if the pin is push and pull capable or has an internal pull-up or pull-down network.)
  - output pins can not be connected to other output pins or bidirectional (avoiding of possible short circuit)
- Certain bus pins (SDA of i2c) , … are bidirectional
  - Any pin that can be in both input or output state depending on current system state.
- Pins for passive devices and pins that are always only connected to other passive devices are passive
- Pins that are intended to supply power to other devices use power output (output pin of a dc/dc converter, a voltage regulator, …)
  - multiple power output pins can not be connected (avoiding of power supply short)
- “Not connected” is used for all pins of the footprint that have no function in the symbol. (You can make them invisible to reduce clutter but don’t stack them. open-source they will open-source each other.)
  - NC pins can still be connected in kicad but there will be an error in ERC. Note that stacked NC pins are connected.
- Open collector is used for open collector or open drain outputs (output pins that need an external pull up)
- Open emitter are used for open emitter or open source outputs (output pins that need an external pull down)
- Tri-state is used for output pins that have a high impedance state (high Z) Such pins have the following possible states.
  - High -> low impedance connection to high signal voltage (in most cases the positive supply voltage)
  - Low -> low impedance connection to low signal voltage (in most case GND)
  - High Z -> no connection to anything. (high impedance state)
- Pins who can be both input or output depending on configuration (a typical GPIO pin) can be either marked as bidirectional or unspecified.

For Reference.
Worry about later.

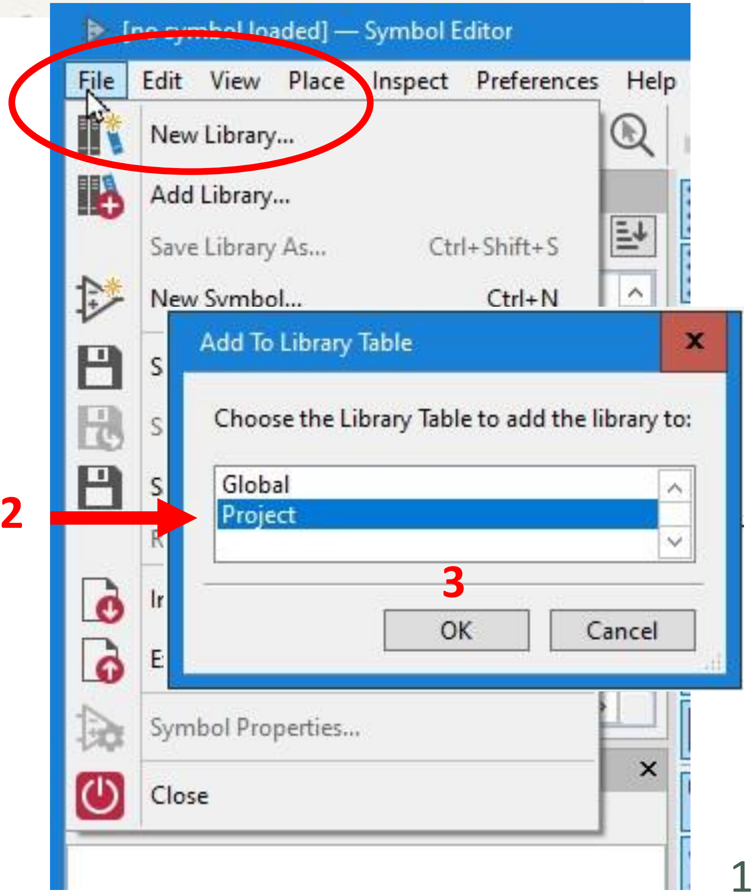
Adding the Symbol to Our Project’s Symbol Library

Symbol Editor


Secret_Robot_Project — Schematic Editor
File Edit View Place Inspect Tools Preferences Help



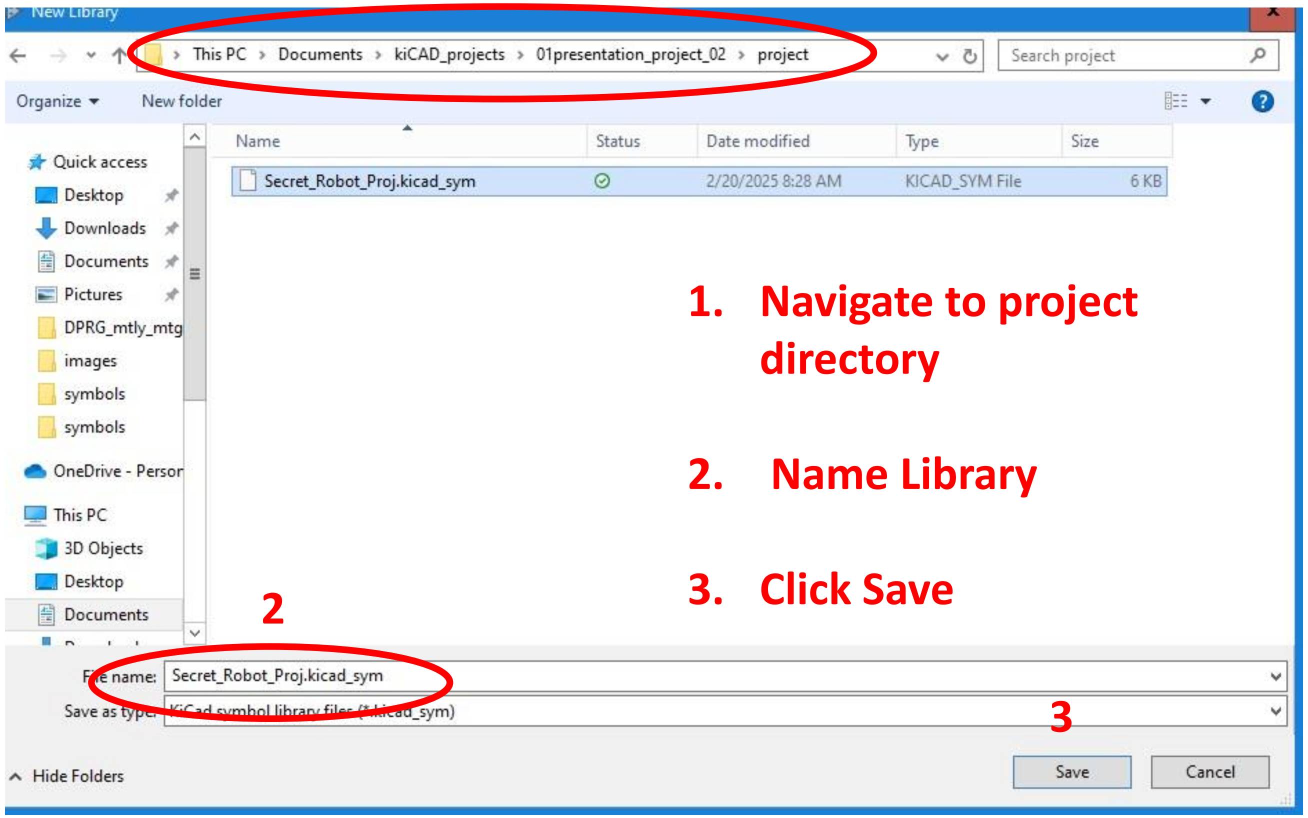
1. File>New Library
2. Select Project
3. Click Ok



Choose the Library Table to add the library to:

Adding the Symbol to Our Project’s Symbol Library

1



New Library

This PC › Documents › kiCAD_projects › 01presentation_project_02 › project

Search project

Name	Status	Date modified	Type	Size
Secret_Robot_Proj.kicad_sym	✓	2/20/2025 8:28 AM	KICAD_SYM File	6 KB

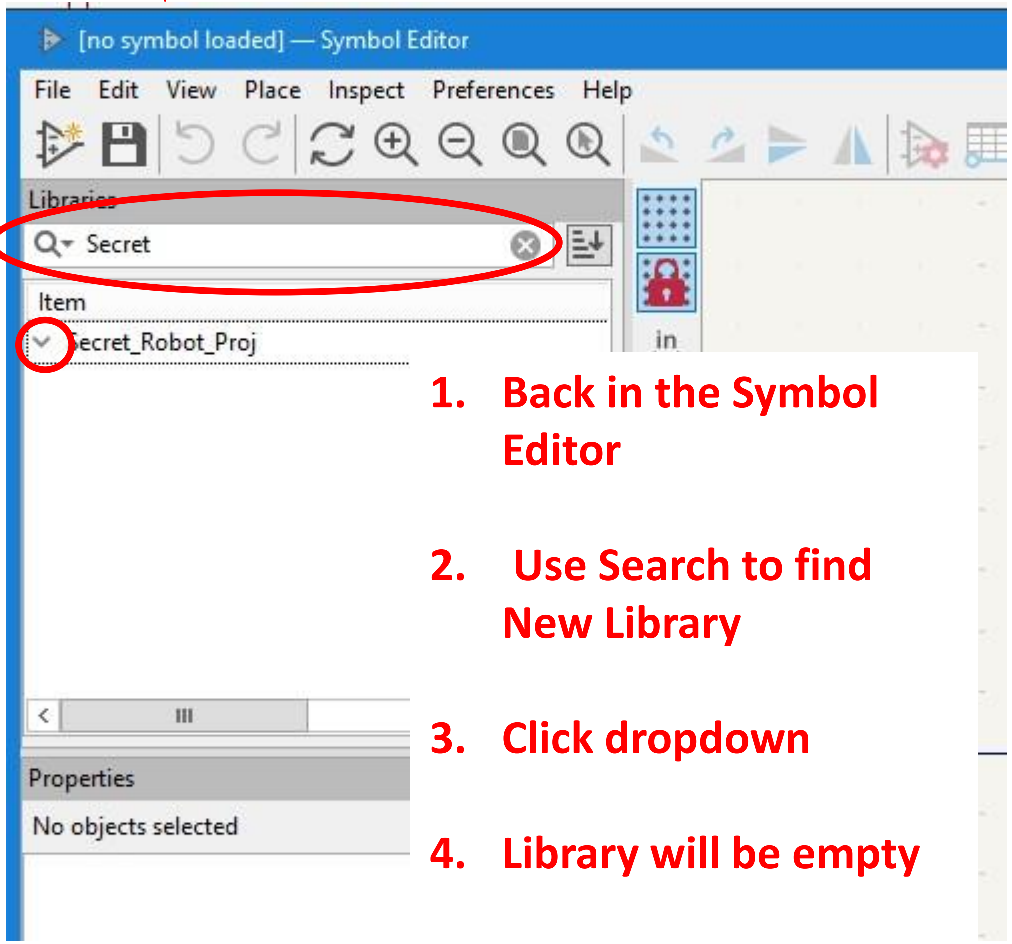
File name: Secret_Robot_Proj.kicad_sym

Save as type: KiCad symbol library files (*.kicad_sym)

1. Navigate to project directory
2. Name Library
3. Click Save

Symbol Editor

1



[no symbol loaded] — Symbol Editor

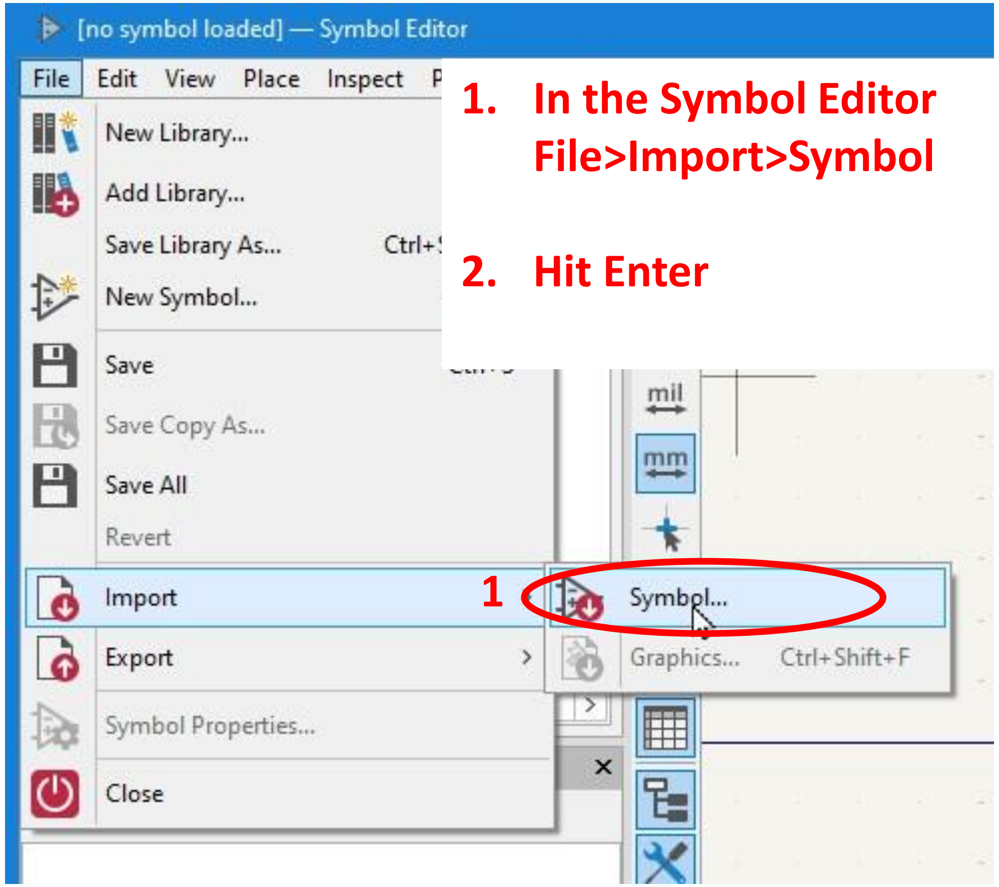
Libraries: Secret

Item: Secret_Robot_Proj

Properties: No objects selected

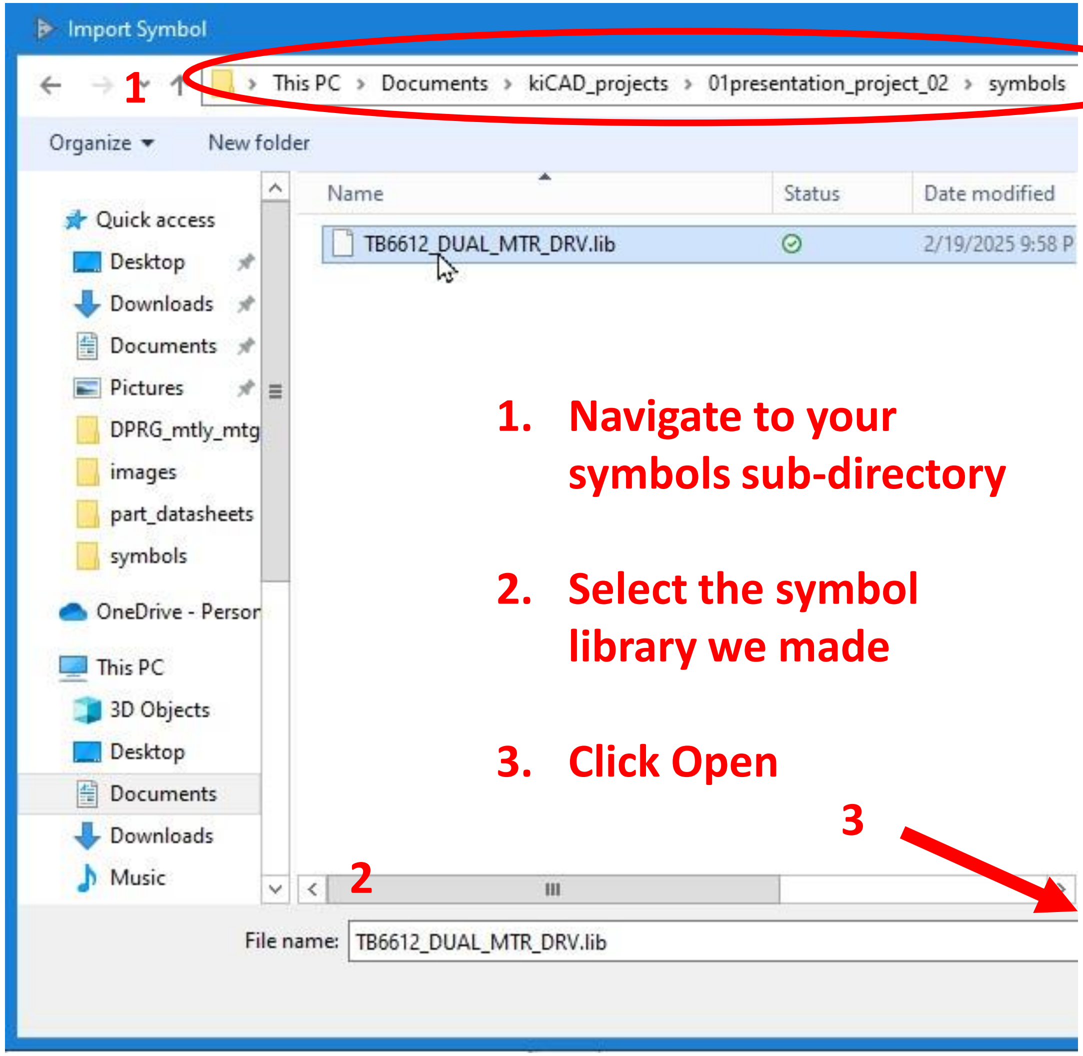
1. Back in the Symbol Editor
2. Use Search to find New Library
3. Click dropdown
4. Library will be empty

Adding the Symbol to Our Project’s Symbol Library



1. In the Symbol Editor File>Import>Symbol
2. Hit Enter

3



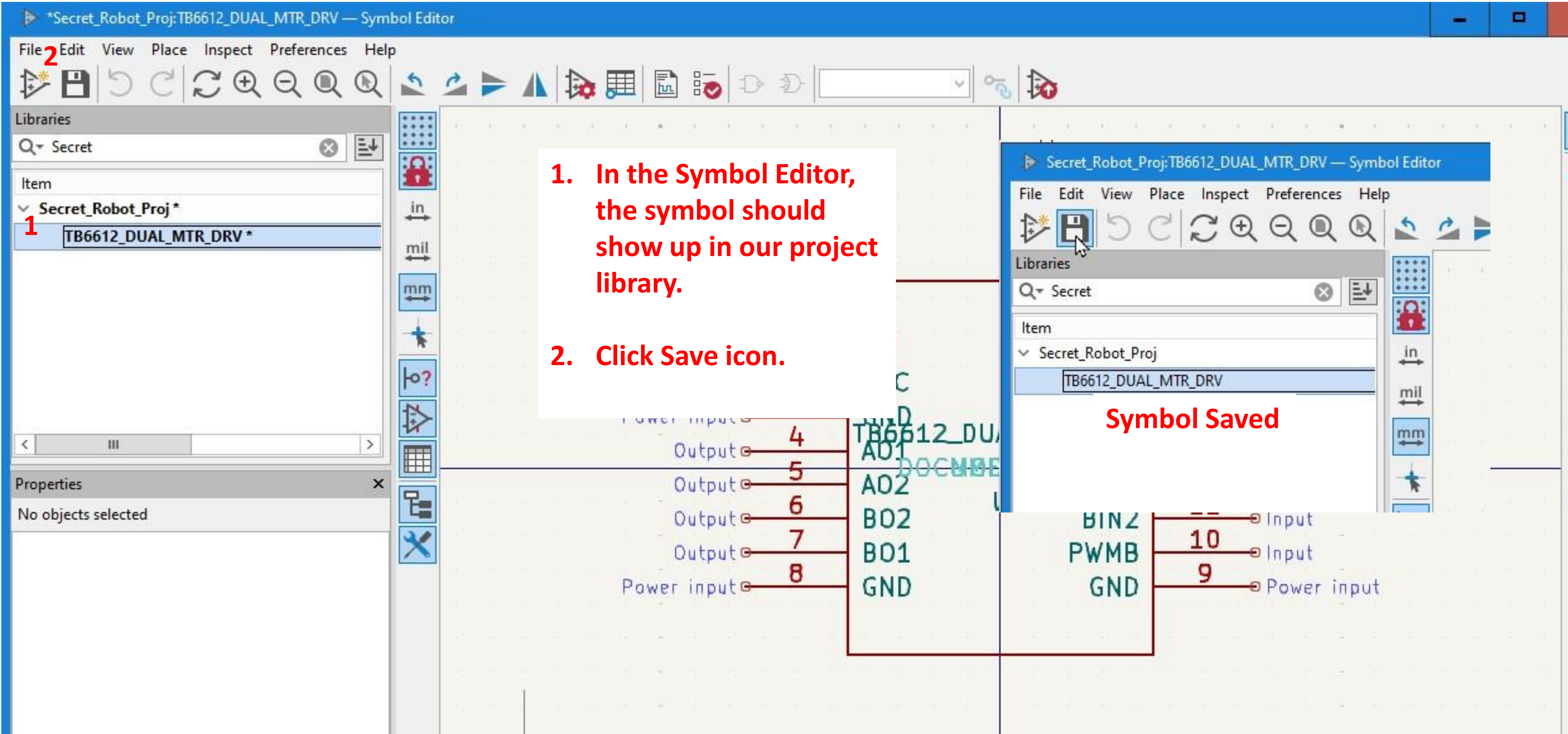
This PC › Documents › kiCAD_projects › 01presentation_project_02 › symbols

TB6612_DUAL_MTR_DRV.lib 2/19/2025 9:58 P

File name: TB6612_DUAL_MTR_DRV.lib

1. Navigate to your symbols sub-directory
2. Select the symbol library we made
3. Click Open

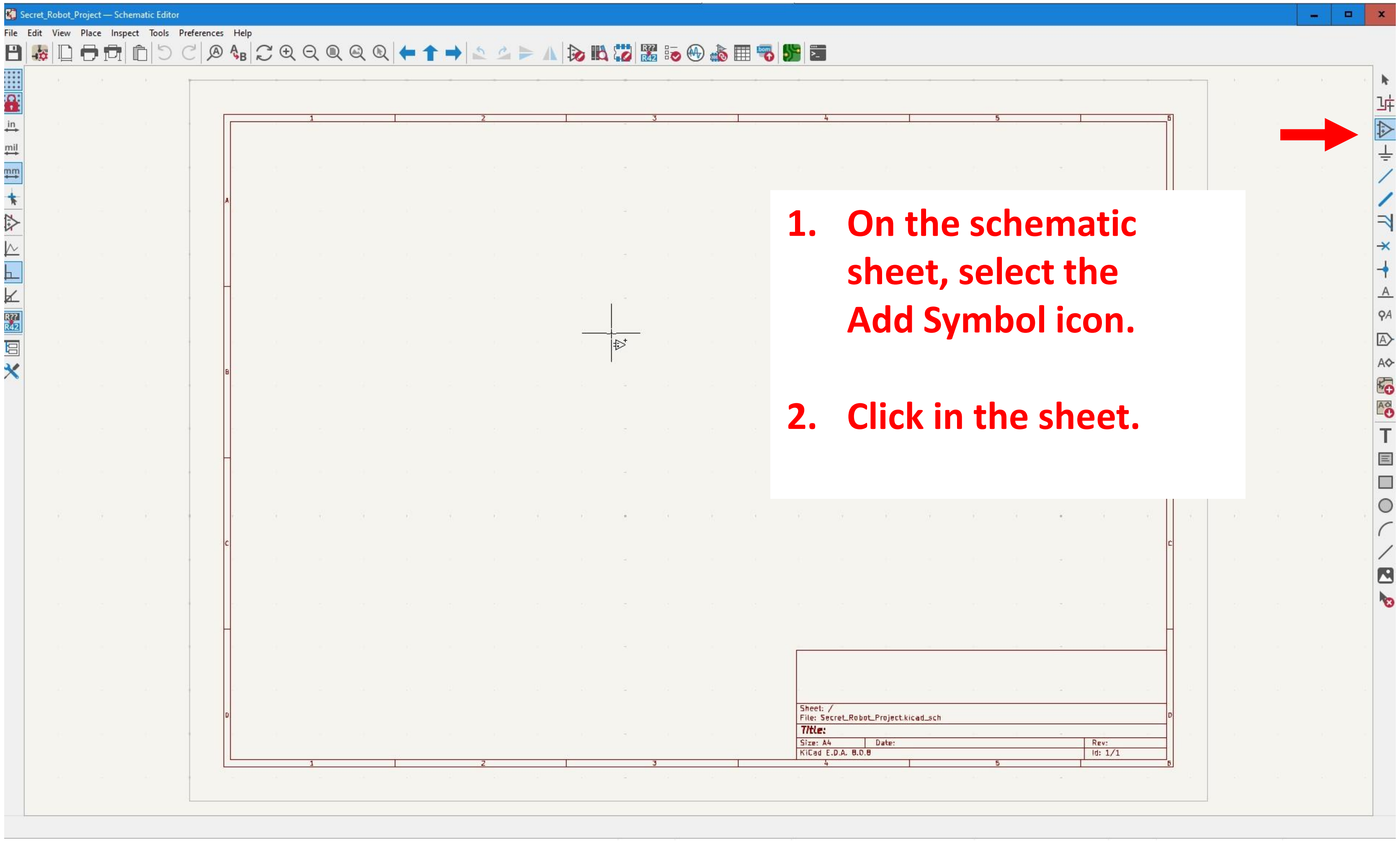
Saving the Symbol We Made



1. In the Symbol Editor, the symbol should show up in our project library.
2. Click Save icon.

Symbol Saved

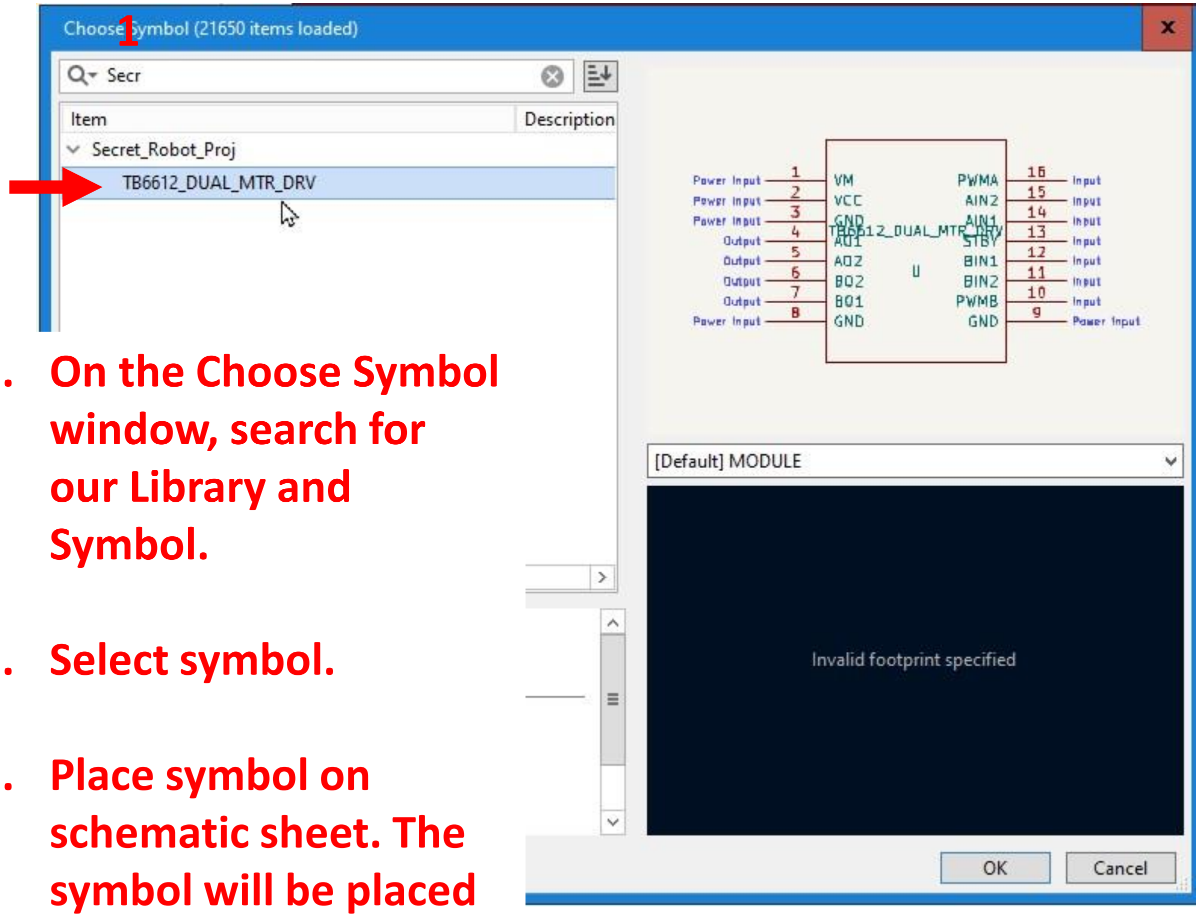
Finally! – Using the Symbol, We Made



1. On the schematic sheet, select the Add Symbol icon.

2. Click in the sheet.

Finally! – Using the Symbol, We Made



Choose Symbol (21650 items loaded)

Secr

Item	Description
Secret_Robot_Proj	
TB6612_DUAL_MTR_DRV	

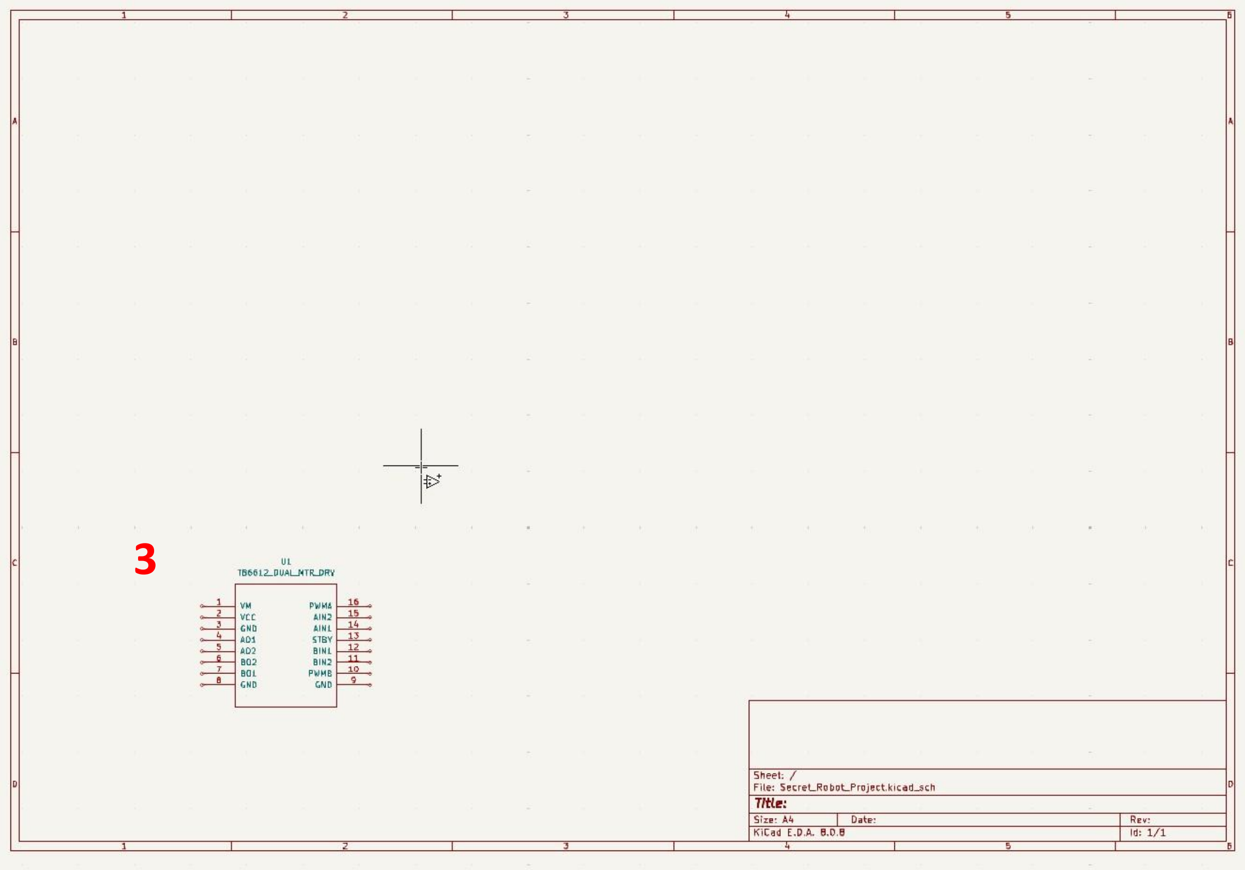
[Default] MODULE

Invalid footprint specified

1. On the Choose Symbol window, search for our Library and Symbol.

2. Select symbol.

3. Place symbol on schematic sheet. The symbol will be placed where you click.



Sheet: /
File: Secret_Robot_Project.kicad_sch
Title:
Size: A4 Date: Rev:
KiCad E.D.A. 8.0.8 Id: 1/1

Symbol Exercise

Place the following symbols on your schematic sheet.

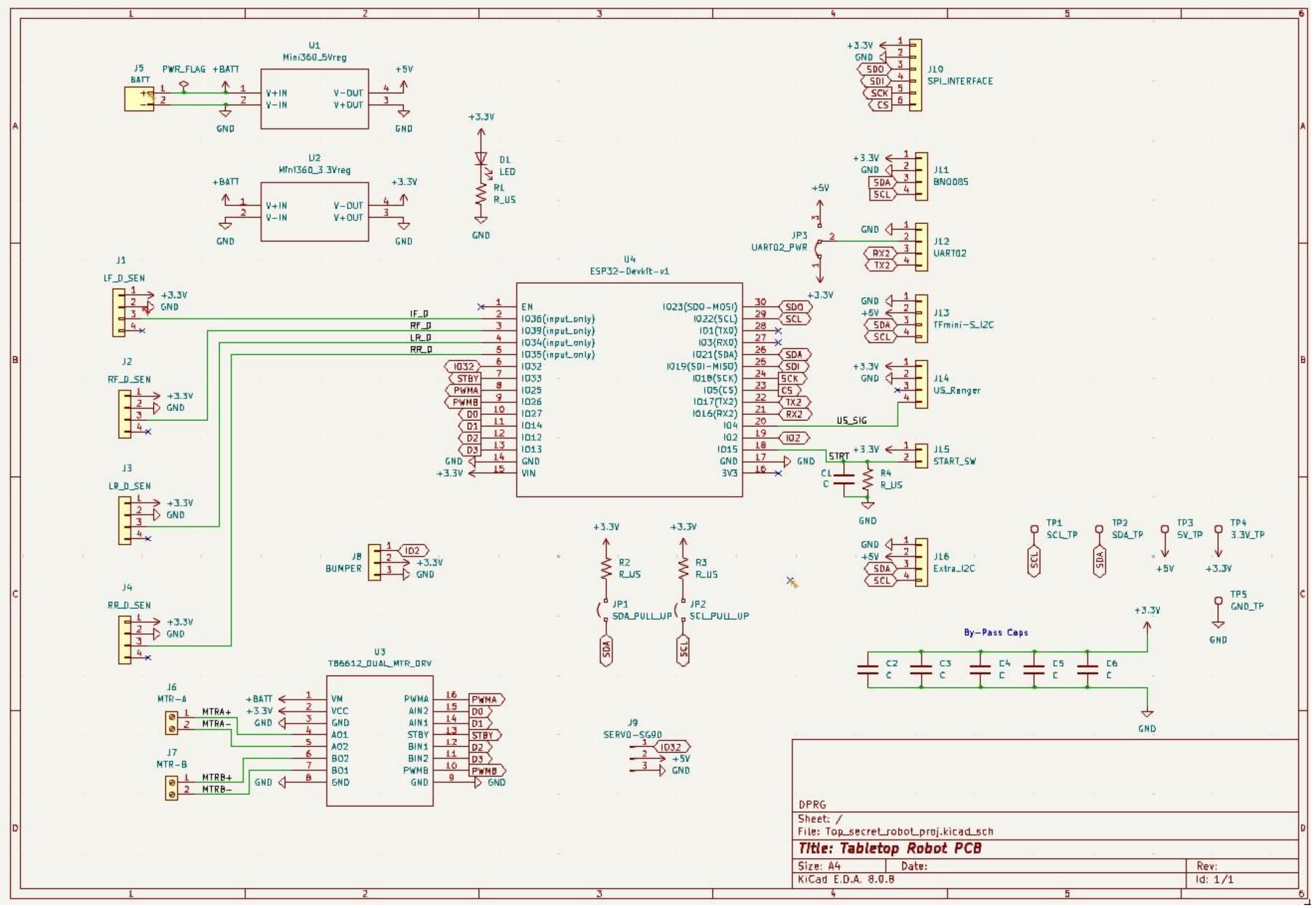
1. **LED**
2. **R_US (resistor using US style symbol)**
3. **Screw_terminal_01x02**
4. **Conn_01x04**
5. **Conn_01x03_Pin**
6. **Jumper_2_Open**
7. **TestPoint**
8. **C (non-polarized capacitor)**
9. **GND**
10. **+3.3V**
11. **PWR_FLAG**

Use the "Add Power" Icon (below the "Add Symbol").

Downloading Some Resources…

- Go to http:// <link to resources> and download the zip file and unzip it.
- Open the project file

The New Schematic



1. Find the ERC icon and run ERC.
2. You should see two warnings and one error.
3. Warnings are marked with a yellow arrow and errors are marked with a red arrow.
4. Fix one warning and the error.

The new project symbol library is "Presentation_Proj_Library".

Exercise Solution…

The two warnings are:

- A modified instance of a connector. This one was done on purpose and is of little consequence.

- An extra “not-connected” symbol in the C-4 area of the sheet. This is a common extra mouse click type of error. This symbol should be removed.

- The error is a missing PWR_FLAG on the “Bminus” trace, which goes to the V-IN pins to both voltage regulators (GND).

PWR_FLAGs

- PWR_FLAGs are used by KiCad as a way to tell the ERC that power nets are connected to a power source when there is no explicit power source.
- The most common situation is an external battery being connected to a voltage regulator through a connector.
- Power Input type pins must be connected to a Power Output type pin. The pins on a connector are Passive.
- PWR_FLAG symbol is found in the Add Power symbols.

Net Classes

Q: What is a “net” vs. a “trace”?

A: A "net" is not the same as a "trace," but rather a "net" represents a group of connected components on a circuit board, while a "trace" is the physical copper line on the PCB that connects those components within a specific net; essentially, a net is the logical connection, and a trace is the physical manifestation of that connection on the PCB.

For example, A net consisting of +3.3V would be all the connections of +3.3V, where a connection between the +3.3V regulator and a sensor would be a trace.

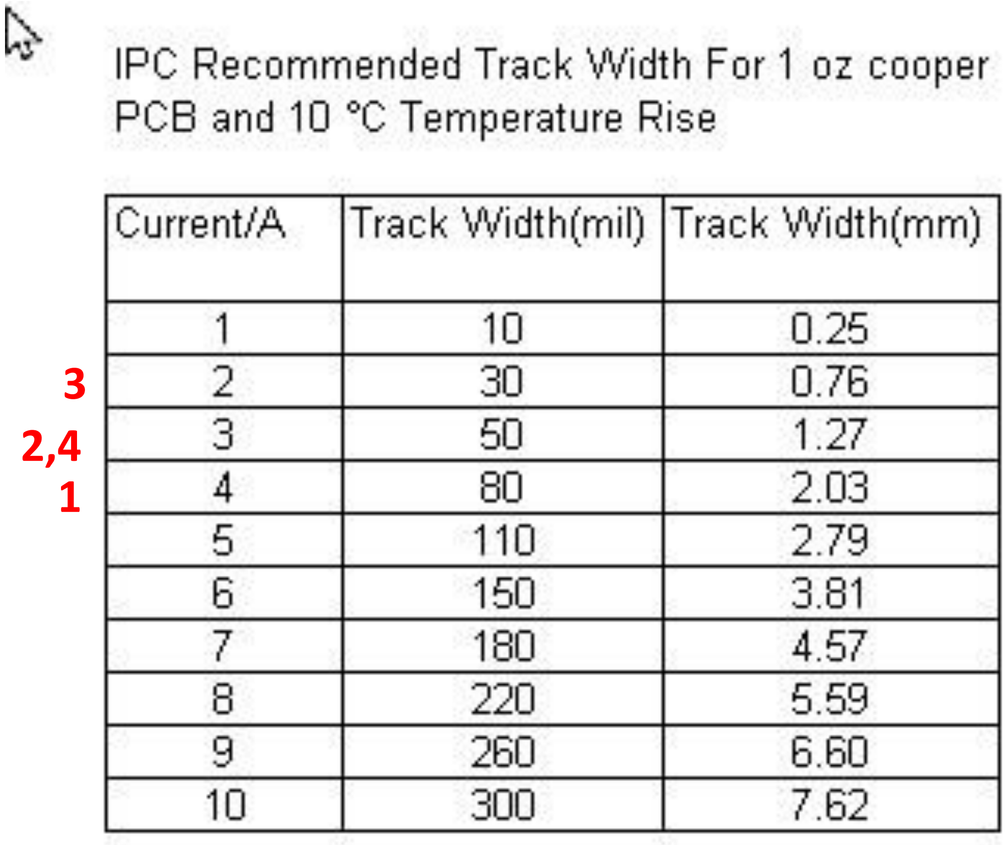
Net Classes

Q: What if I want different traces to be different design rules?

A: Netclasses

Net Classes are groups of nets that can be assigned design rules (for the PCB) and graphical properties (for the schematic). In KiCad, each net is part of exactly one net class. If you do not add a net to a specific class, it will be part of the Default class, which always exists.

Trace Width vs. Current



IPC Recommended Track Width For 1 oz cooper PCB and 10 °C Temperature Rise

	Current/A	Track Width(mil)	Track Width(mm)
	1	10	0.25
3	2	30	0.76
2,4	3	50	1.27
1	4	80	2.03
	5	110	2.79
	6	150	3.81
	7	180	4.57
	8	220	5.59
	9	260	6.60
	10	300	7.62

We need to add a few netclasses:

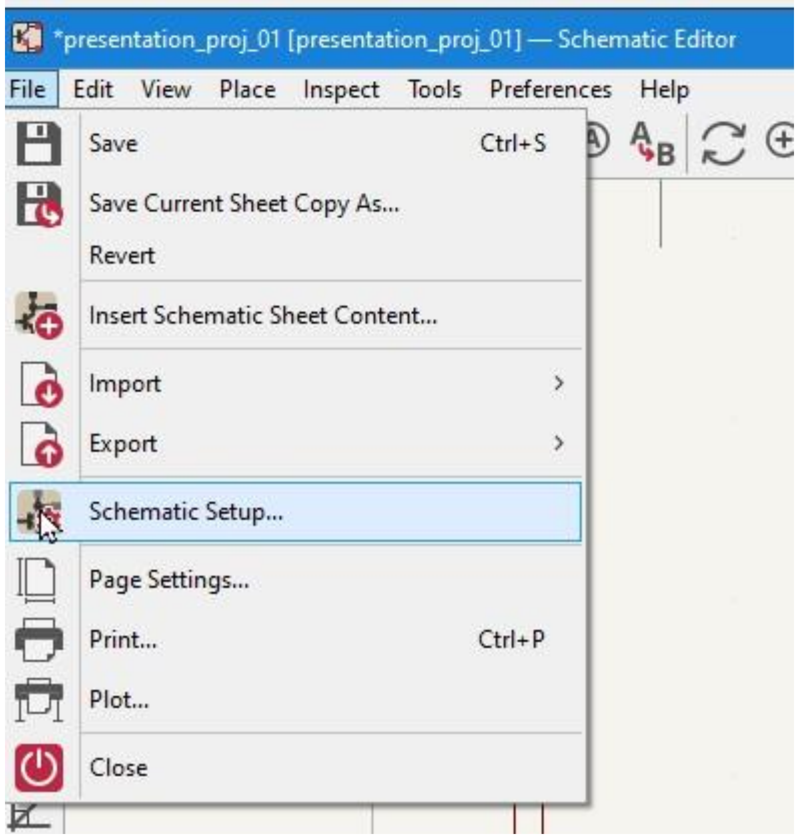
1. **Battery traces (PCB and Mtr Controller)**
2. **Motor power outputs**
3. **3.3V and 5V**
4. **GND**

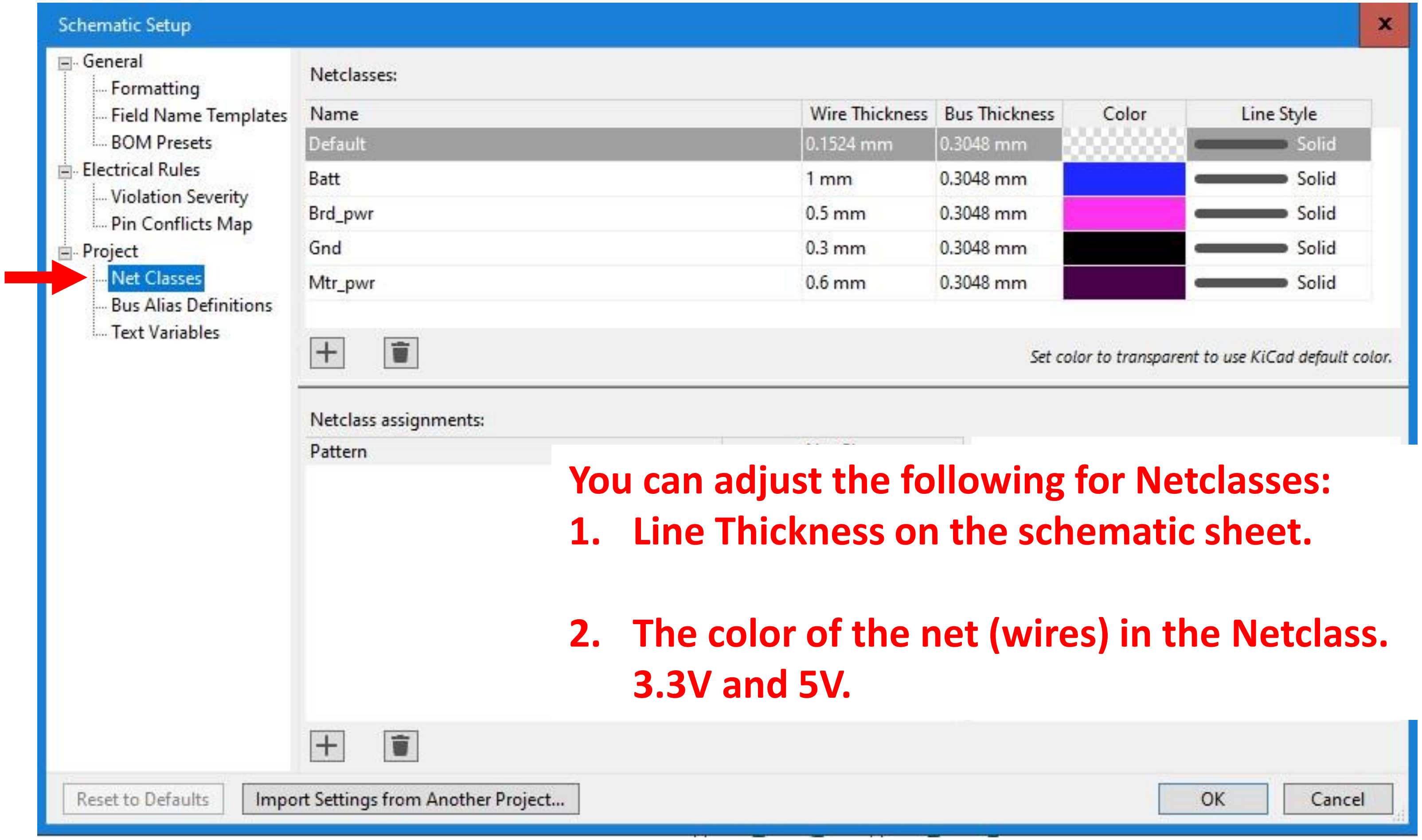
All nets start in the default netclass, which has a minimum trace width of 10mils or 0.25mm.

You should also consider minimum spaces also.

Setting Up New Net Classes

Schematic Graphical Properties





Netclasses:

Name	Wire Thickness	Bus Thickness	Color	Line Style
Default	0.1524 mm	0.3048 mm	▨	Solid
Batt	1 mm	0.3048 mm	🟦	Solid
Brd_pwr	0.5 mm	0.3048 mm	🟪	Solid
Gnd	0.3 mm	0.3048 mm	⬛	Solid
Mtr_pwr	0.6 mm	0.3048 mm	🟪	Solid

Set color to transparent to use KiCad default color.

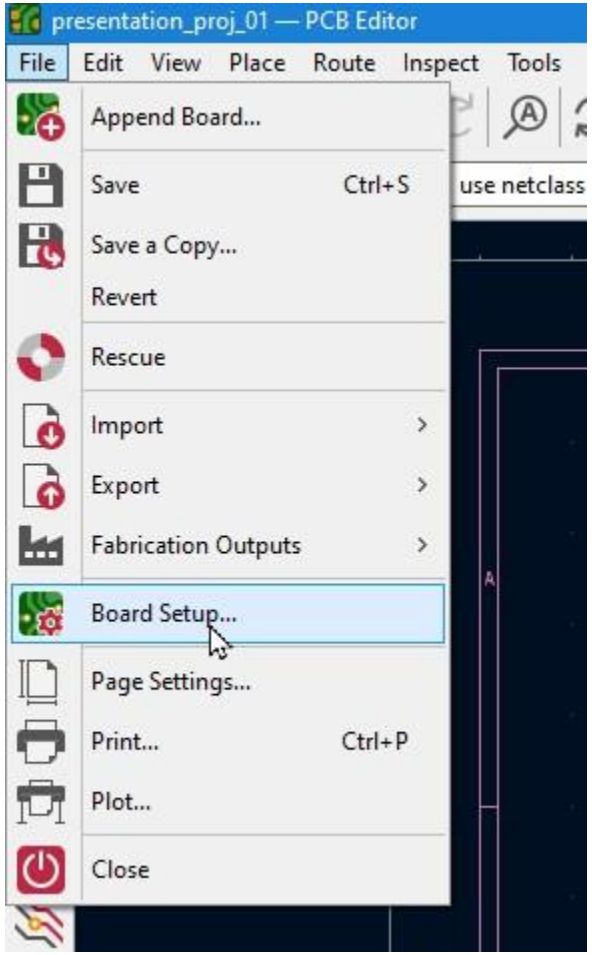
You can adjust the following for Netclasses:

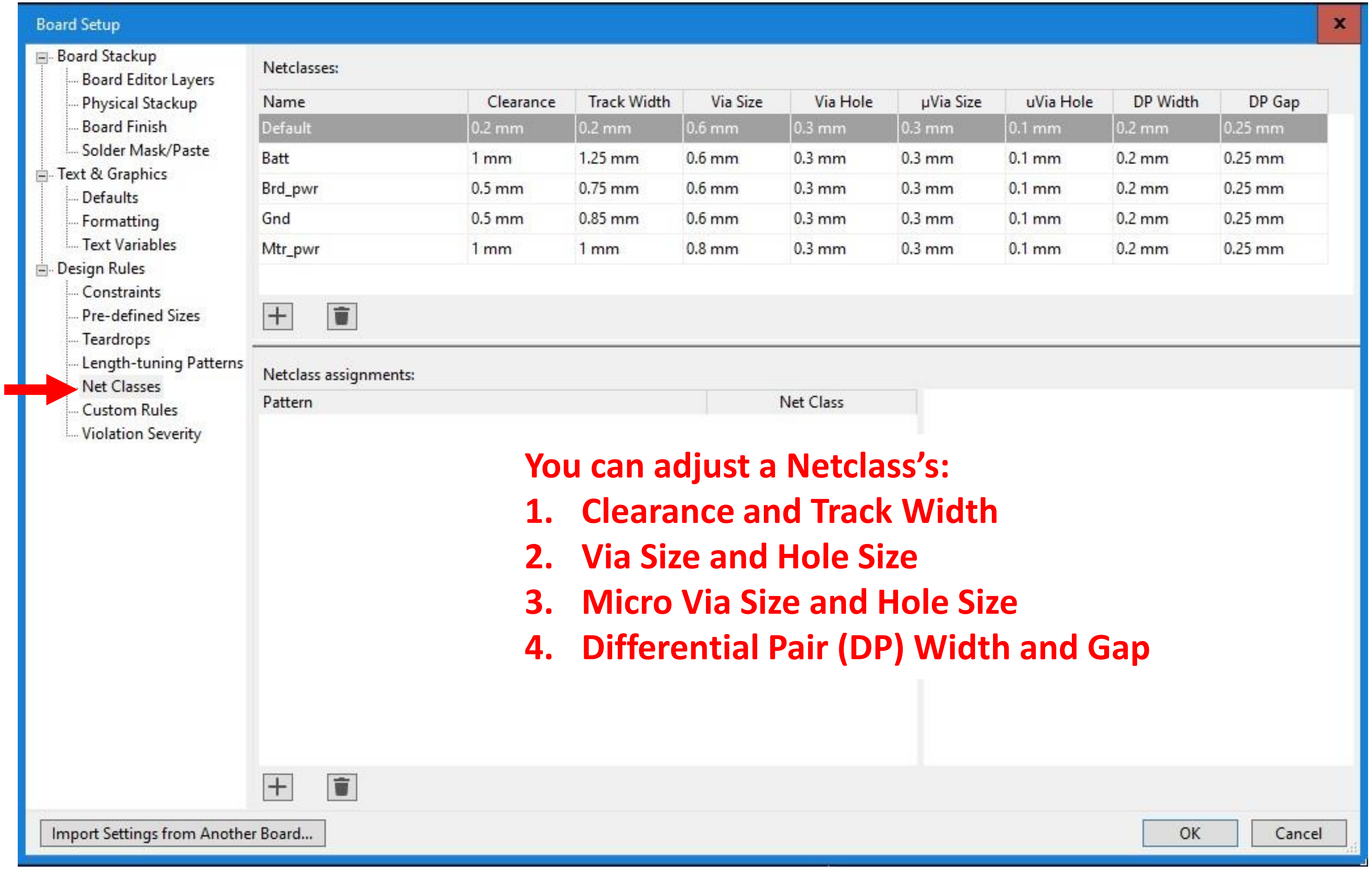
1. **Line Thickness on the schematic sheet.**

2. **The color of the net (wires) in the Netclass. 3.3V and 5V.**

Setting Up New Net Classes

PCB Design Rules





Netclasses:

Name	Clearance	Track Width	Via Size	Via Hole	µVia Size	uVia Hole	DP Width	DP Gap
Default	0.2 mm	0.2 mm	0.6 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm
Batt	1 mm	1.25 mm	0.6 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm
Brd_pwr	0.5 mm	0.75 mm	0.6 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm
Gnd	0.5 mm	0.85 mm	0.6 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm
Mtr_pwr	1 mm	1 mm	0.8 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm

Netclass assignments:

Pattern	Net Class

You can adjust a Netclass's:

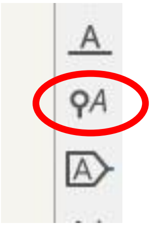
1. **Clearance and Track Width**
2. **Via Size and Hole Size**
3. **Micro Via Size and Hole Size**
4. **Differential Pair (DP) Width and Gap**

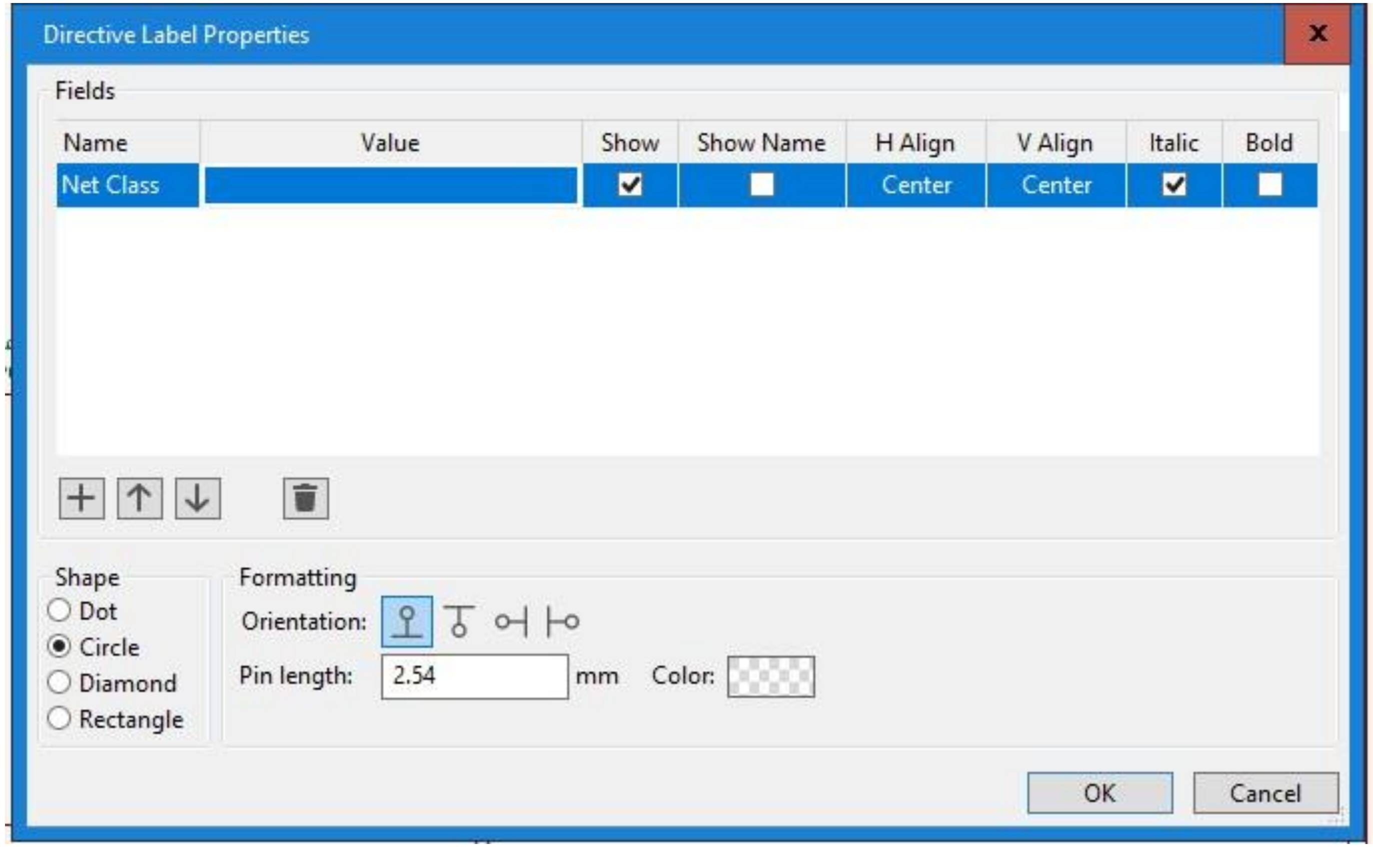
How to Put Nets into Net Classes…

After you setup your Net Classes, you can assign nets to them.

First Way – Add Net Class Directive Labels

Right-side Menu





Directive Label Properties

Fields

Name	Value	Show	Show Name	H Align	V Align	Italic	Bold
Net Class		☑	□	Center	Center	☑	□

Shape
- ○ Dot
- ● Circle
- ○ Diamond
- ○ Rectangle

Formatting

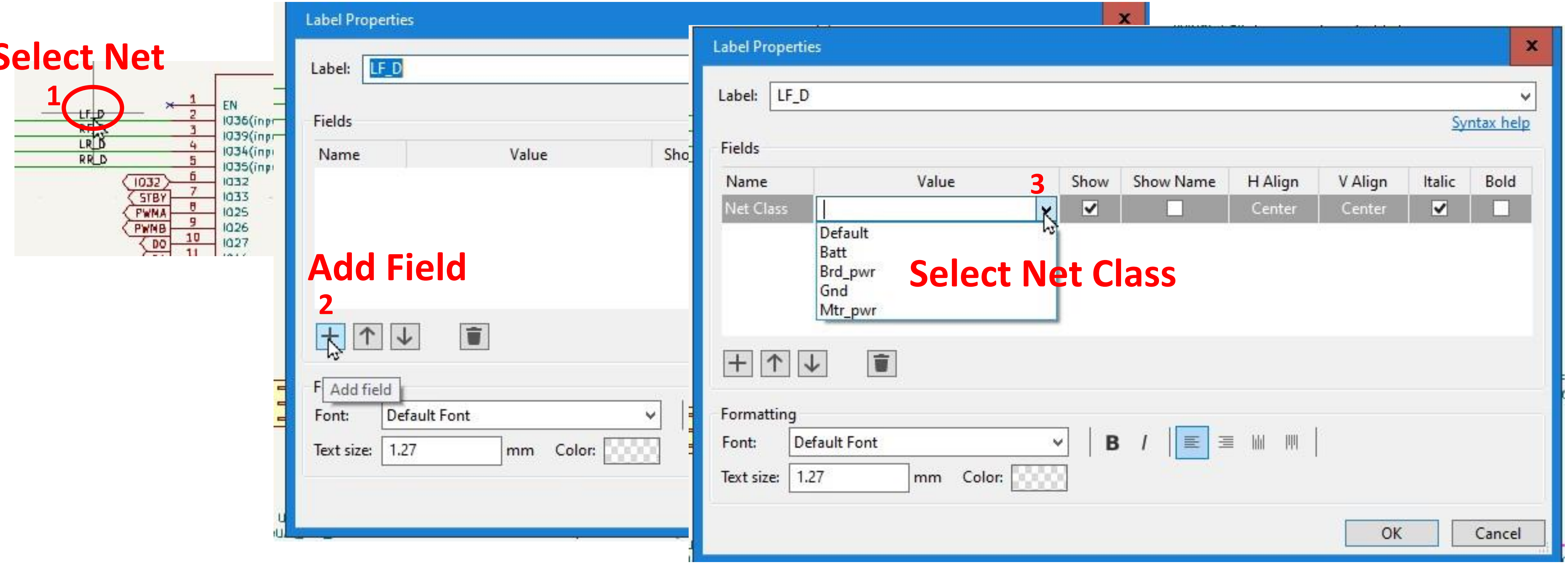
Orientation:

Pin length: 2.54 mm Color:

OK Cancel

How to Put Nets into Net Classes…

Second Way – Click on Net and add Net Class Field



Select Net

1

Add Field

2

3

Select Net Class

Label Properties

Label: LF_D

Fields

Name	Value	Sho

Add field

Font: Default Font

Text size: 1.27 mm Color:

Label Properties

Label: LF_D

Syntax help

Fields

Name	Value	Show	Show Name	H Align	V Align	Italic	Bold
Net Class		☑	□	Center	Center	☑	□

Default
Batt
Brd_pwr
Gnd
Mtr_pwr

Formatting

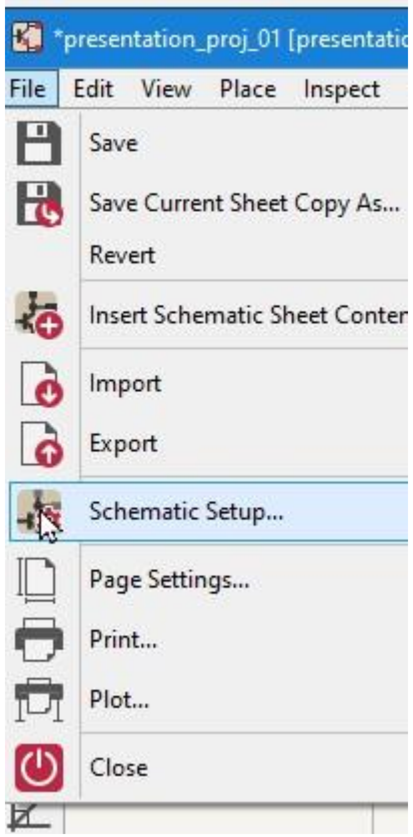
Font: Default Font

Text size: 1.27 mm Color:

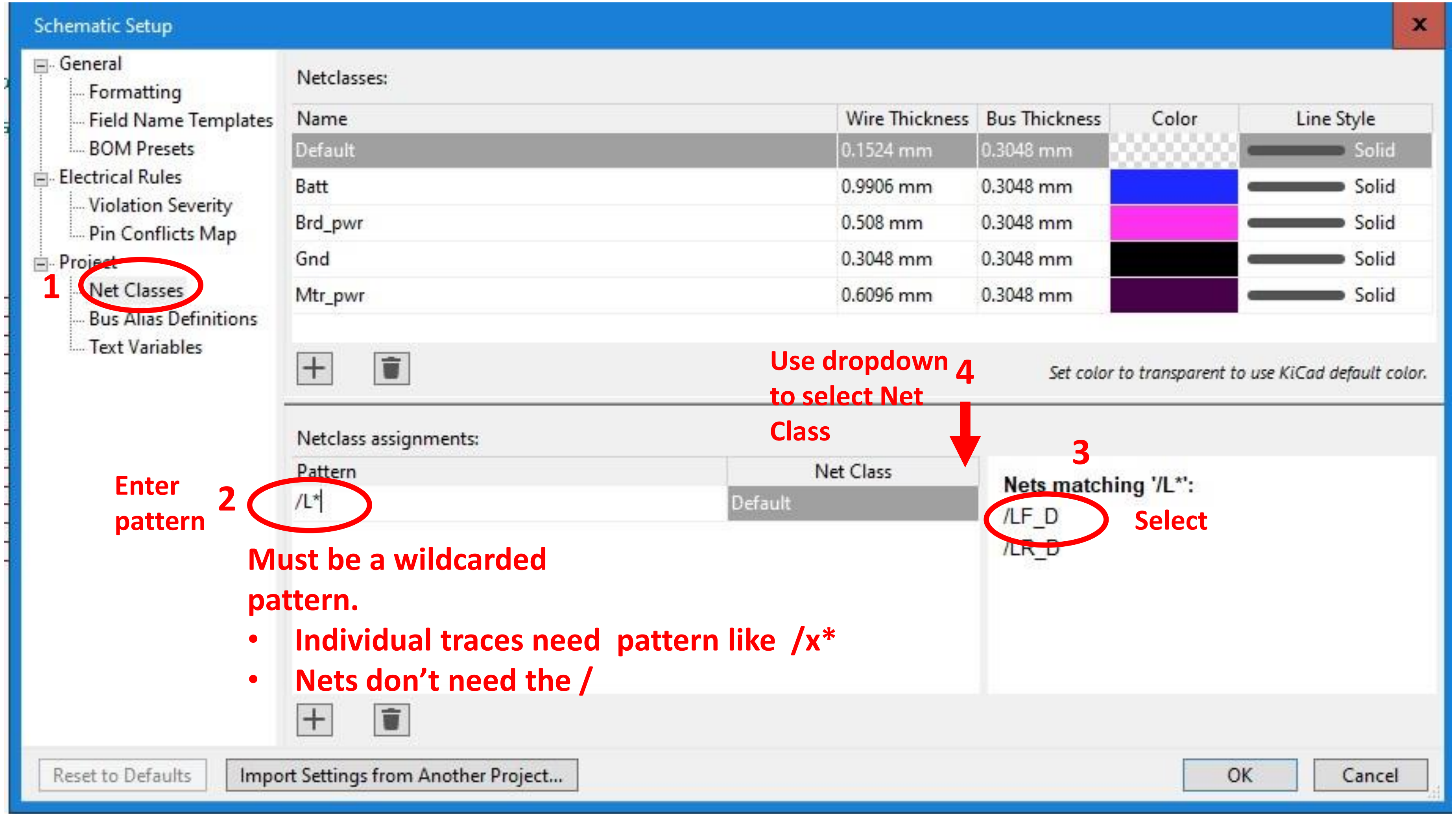
OK Cancel

How to Put Nets into Net Classes…

Third Way – Use the Schematic or PCB setup window



*presentation_proj_01 [presentati



Schematic Setup

Netclasses:

Name	Wire Thickness	Bus Thickness	Color	Line Style
Default	0.1524 mm	0.3048 mm	▨	Solid
Batt	0.9906 mm	0.3048 mm	🟦	Solid
Brd_pwr	0.508 mm	0.3048 mm	🟪	Solid
Gnd	0.3048 mm	0.3048 mm	⬛	Solid
Mtr_pwr	0.6096 mm	0.3048 mm	🟪	Solid

Set color to transparent to use KiCad default color.

Netclass assignments:

Pattern	Net Class
/L*	Default

Nets matching '/L*':

/LF_D

/LR_D

1

Enter pattern 2

3 Select

Use dropdown to select Net Class 4

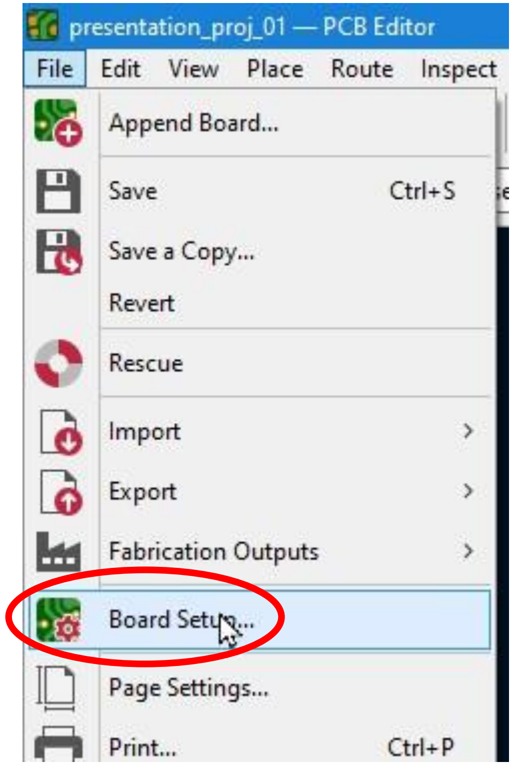
Must be a wildcarded pattern.

- Individual traces need pattern like /x*
- Nets don't need the /

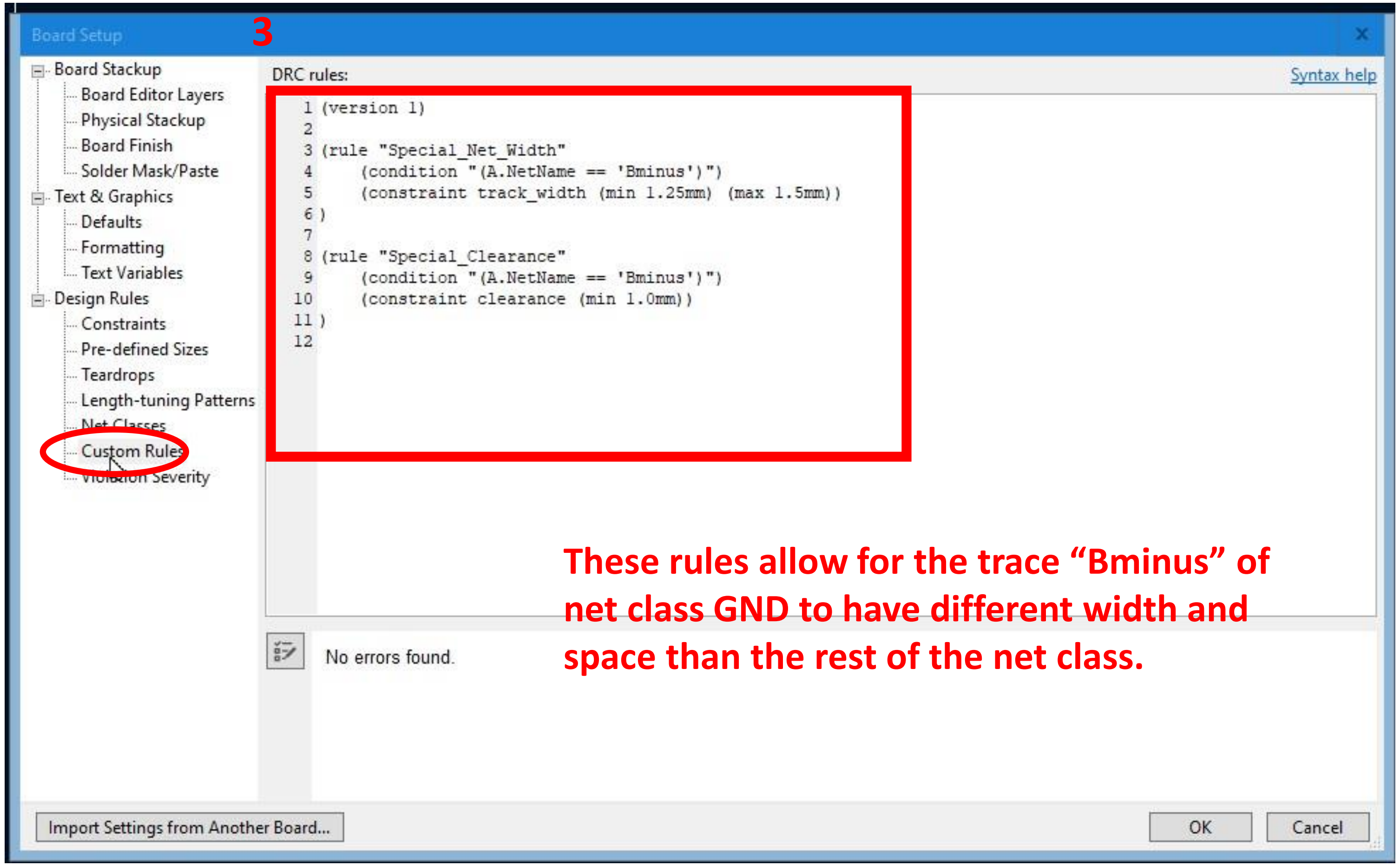
How to Put Nets into Net Classes…

Fourth Way – For Special Cases, Use Custom Rules in the PCB setup window

1



2



```
(version 1)

(rule "Special_Net_Width"
    (condition "(A.NetName == 'Bminus')")
    (constraint track_width (min 1.25mm) (max 1.5mm))
)

(rule "Special_Clearance"
    (condition "(A.NetName == 'Bminus')")
    (constraint clearance (min 1.0mm))
)

```

No errors found.

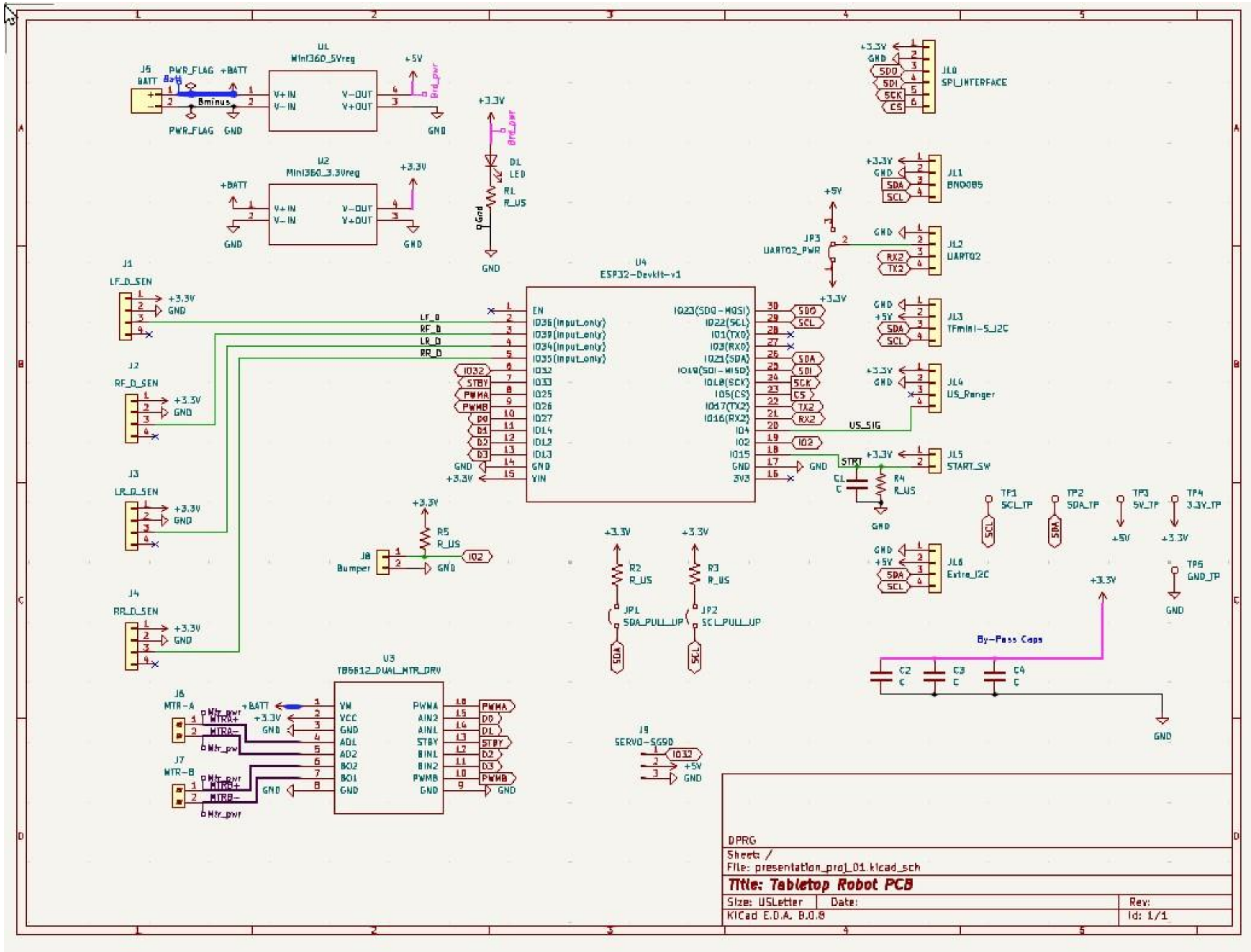
These rules allow for the trace “Bminus” of net class GND to have different width and space than the rest of the net class.

Before Starting…

It is a good idea to gather all the parts you are going to use.

It makes it easier later when we start to select footprints.

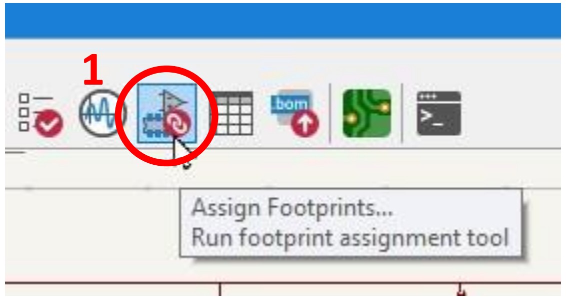
Final Schematic Sheet



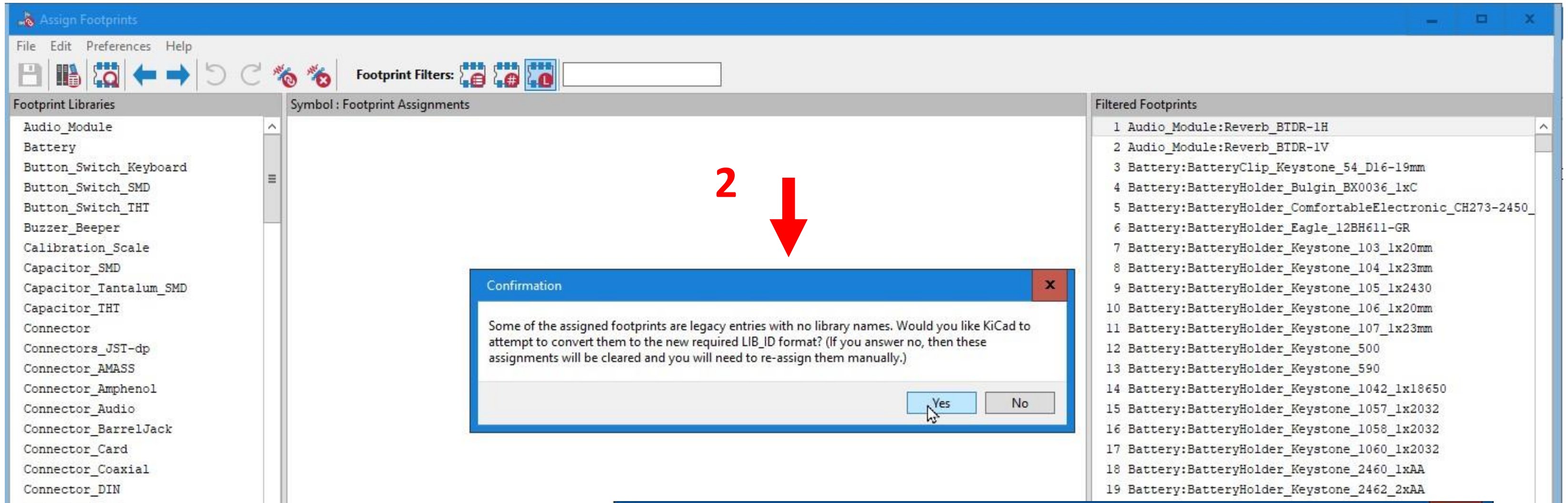
Footprints

Before you can layout a PCB, you need to have footprints for every part.

Footprint Assignment Tool



Assign Footprints...
Run footprint assignment tool



Footprint Libraries

```
Audio_Module
Battery
Button_Switch_Keyboard
Button_Switch_SMD
Button_Switch_THT
Buzzer_Beeper
Calibration_Scale
Capacitor_SMD
Capacitor_Tantalum_SMD
Capacitor_THT
Connector
Connectors_JST-dp
Connector_AMASS
Connector_Amphenol
Connector_Audio
Connector_BarrelJack
Connector_Card
Connector_Coaxial
Connector_DIN
Connector_Dsub
Connector_FFC-FPC
```

Symbol : Footprint Assignments

Filtered Footprints

```
 1 Audio_Module:Reverb_BTDR-1H
 2 Audio_Module:Reverb_BTDR-1V
 3 Battery:BatteryClip_Keystone_54_D16-19mm
 4 Battery:BatteryHolder_Bulgin_BX0036_1xC
 5 Battery:BatteryHolder_ComfortableElectronic_CH273-2450_
 6 Battery:BatteryHolder_Eagle_12BH611-GR
 7 Battery:BatteryHolder_Keystone_103_1x20mm
 8 Battery:BatteryHolder_Keystone_104_1x23mm
 9 Battery:BatteryHolder_Keystone_105_1x2430
10 Battery:BatteryHolder_Keystone_106_1x20mm
11 Battery:BatteryHolder_Keystone_107_1x23mm
12 Battery:BatteryHolder_Keystone_500
13 Battery:BatteryHolder_Keystone_590
14 Battery:BatteryHolder_Keystone_1042_1x18650
15 Battery:BatteryHolder_Keystone_1057_1x2032
16 Battery:BatteryHolder_Keystone_1058_1x2032
17 Battery:BatteryHolder_Keystone_1060_1x2032
18 Battery:BatteryHolder_Keystone_2460_1xAA
19 Battery:BatteryHolder_Keystone_2462_2xAA
```

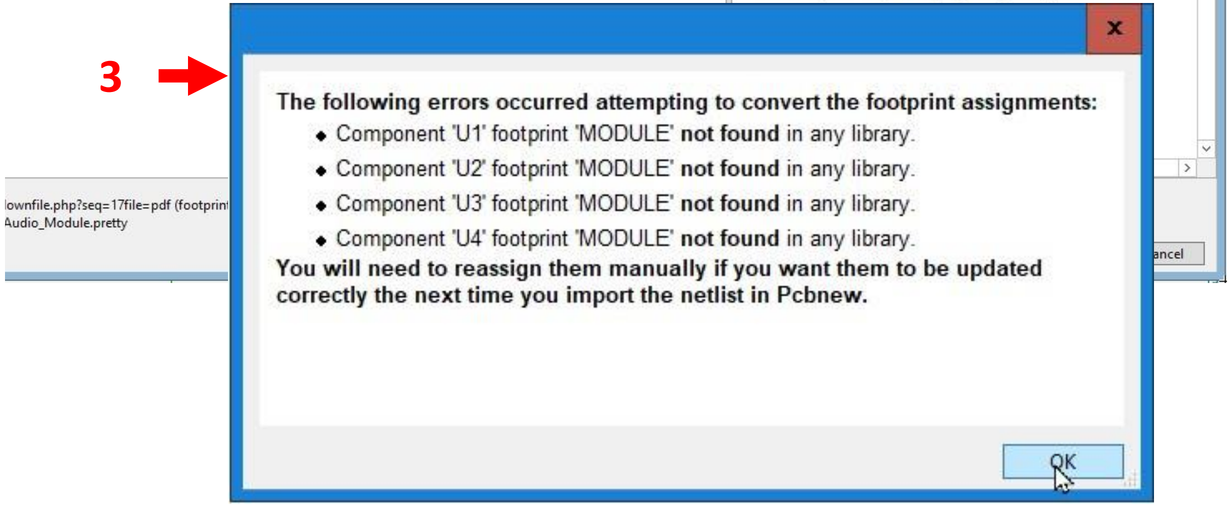
Confirmation

Some of the assigned footprints are legacy entries with no library names. Would you like KiCad to attempt to convert them to the new required LIB_ID format? (If you answer no, then these assignments will be cleared and you will need to re-assign them manually.)

When opening the Footprint Assignment Tool You might see some error boxes. Just click "yes" and OK.

1. Footprint Assignment tool
2. Asking if you want to update footprint formats (KiCad is always evolving)
3. Essentially says that KiCad can't find footprints.

3

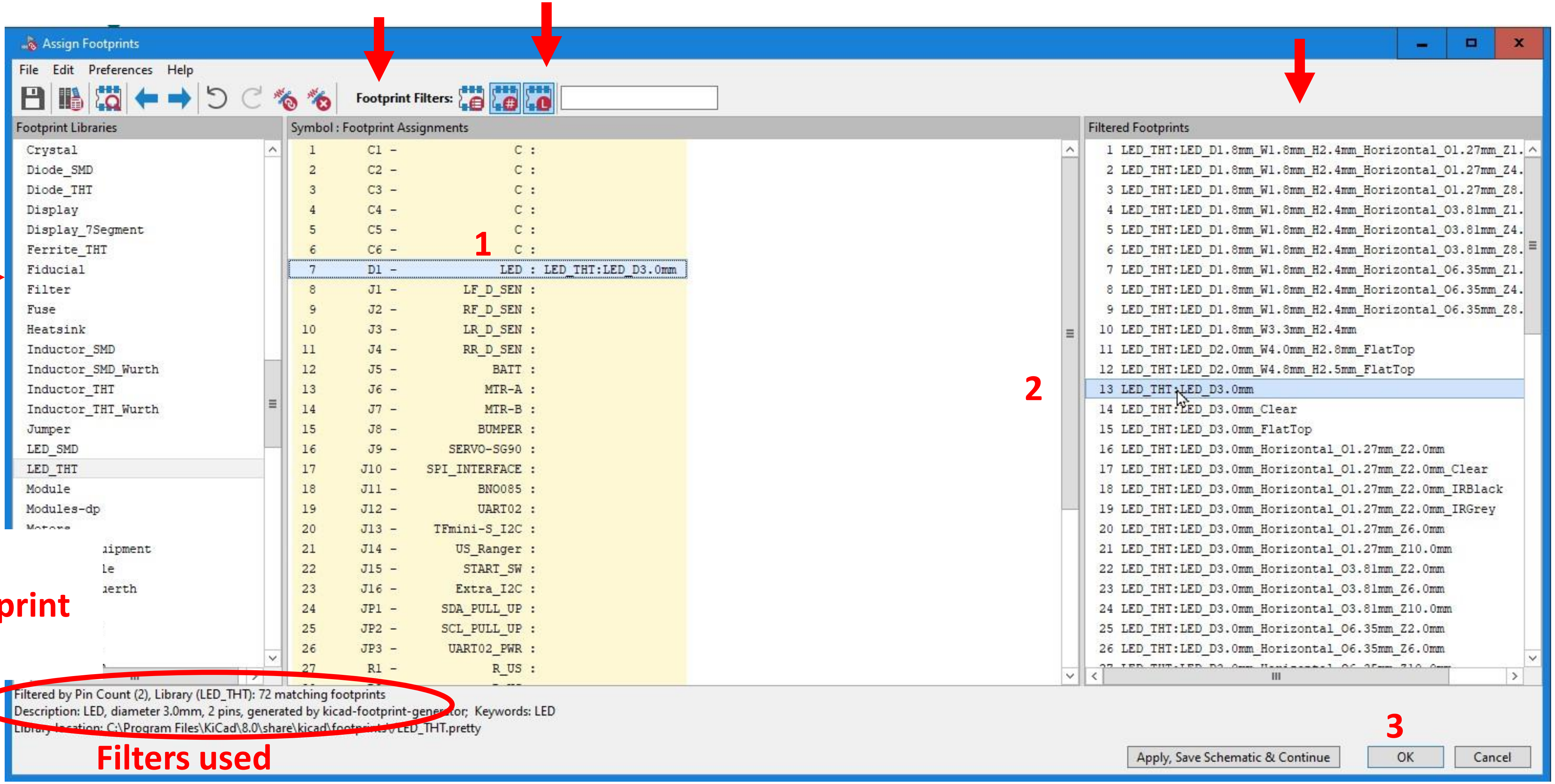


The following errors occurred attempting to convert the footprint assignments:

- Component 'U1' footprint 'MODULE' **not found** in any library.
- Component 'U2' footprint 'MODULE' **not found** in any library.
- Component 'U3' footprint 'MODULE' **not found** in any library.
- Component 'U4' footprint 'MODULE' **not found** in any library.

You will need to reassign them manually if you want them to be updated correctly the next time you import the netlist in Pcbnew.

Assigning a Footprint



Footprint Libraries

```
Crystal
Diode_SMD
Diode_THT
Display
Display_7Segment
Ferrite_THT
Fiducial
Filter
Fuse
Heatsink
Inductor_SMD
Inductor_SMD_Wurth
Inductor_THT
Inductor_THT_Wurth
Jumper
LED_SMD
LED_THT
Module
Modules-dp
Motors
uipment
le
uerth
```

Symbol : Footprint Assignments

```
 1    C1 -             C :
 2    C2 -             C :
 3    C3 -             C :
 4    C4 -             C :
 5    C5 -             C :
 6    C6 -             C :
 7    D1 -           LED : LED_THT:LED_D3.0mm
 8    J1 -      LF_D_SEN :
 9    J2 -      RF_D_SEN :
10    J3 -      LR_D_SEN :
11    J4 -      RR_D_SEN :
12    J5 -          BATT :
13    J6 -         MTR-A :
14    J7 -         MTR-B :
15    J8 -        BUMPER :
16    J9 -     SERVO-SG90 :
17   J10 - SPI_INTERFACE :
18   J11 -        BNO085 :
19   J12 -        UART02 :
20   J13 -   TFmini-S_I2C :
21   J14 -     US_Ranger :
22   J15 -      START_SW :
23   J16 -     Extra_I2C :
24   JP1 -   SDA_PULL_UP :
25   JP2 -   SCL_PULL_UP :
26   JP3 -    UART02_PWR :
27    R1 -          R_US :
```

Filtered Footprints

```
 1 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O1.27mm_Z1.
 2 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O1.27mm_Z4.
 3 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O1.27mm_Z8.
 4 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O3.81mm_Z1.
 5 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O3.81mm_Z4.
 6 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O3.81mm_Z8.
 7 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O6.35mm_Z1.
 8 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O6.35mm_Z4.
 9 LED_THT:LED_D1.8mm_W1.8mm_H2.4mm_Horizontal_O6.35mm_Z8.
10 LED_THT:LED_D1.8mm_W3.3mm_H2.4mm
11 LED_THT:LED_D2.0mm_W4.0mm_H2.8mm_FlatTop
12 LED_THT:LED_D2.0mm_W4.8mm_H2.5mm_FlatTop
13 LED_THT:LED_D3.0mm
14 LED_THT:LED_D3.0mm_Clear
15 LED_THT:LED_D3.0mm_FlatTop
16 LED_THT:LED_D3.0mm_Horizontal_O1.27mm_Z2.0mm
17 LED_THT:LED_D3.0mm_Horizontal_O1.27mm_Z2.0mm_Clear
18 LED_THT:LED_D3.0mm_Horizontal_O1.27mm_Z2.0mm_IRBlack
19 LED_THT:LED_D3.0mm_Horizontal_O1.27mm_Z2.0mm_IRGrey
20 LED_THT:LED_D3.0mm_Horizontal_O1.27mm_Z6.0mm
21 LED_THT:LED_D3.0mm_Horizontal_O1.27mm_Z10.0mm
22 LED_THT:LED_D3.0mm_Horizontal_O3.81mm_Z2.0mm
23 LED_THT:LED_D3.0mm_Horizontal_O3.81mm_Z6.0mm
24 LED_THT:LED_D3.0mm_Horizontal_O3.81mm_Z10.0mm
25 LED_THT:LED_D3.0mm_Horizontal_O6.35mm_Z2.0mm
26 LED_THT:LED_D3.0mm_Horizontal_O6.35mm_Z6.0mm
```

```
Filtered by Pin Count (2), Library (LED_THT): 72 matching footprints
Description: LED, diameter 3.0mm, 2 pins, generated by kicad-footprint-generator;  Keywords: LED
Library location: C:\Program Files\KiCad\8.0\share\kicad\footprints\LED_THT.pretty
```

1. Select part
2. Select footprint
3. Click OK

Missing Footprints…

We are lucky because our board has mostly standard parts
Like connectors, resistors, LEDs, and capacitors.

The modules we are using will require finding or building a footprint in the Footprint Editor. Luckily, we found everything.

Since our modules are 2.54mm spaced pins, and we can build our footprints using connectors.

Footprint Assignment Exercise

Assign a footprint for C1 through C6 (all the same) and R1 through R4 (all the same).

Capacitor details: THT, 2 lead, pitch is 5mm, diameter is 4.7, and the width is 2.5mm.

Resistor details: THT, 2 lead, pitch between leads is 10.1mm, the body diameter is 2.5mm, and the body length is 6.3mm.

Exercise Answers

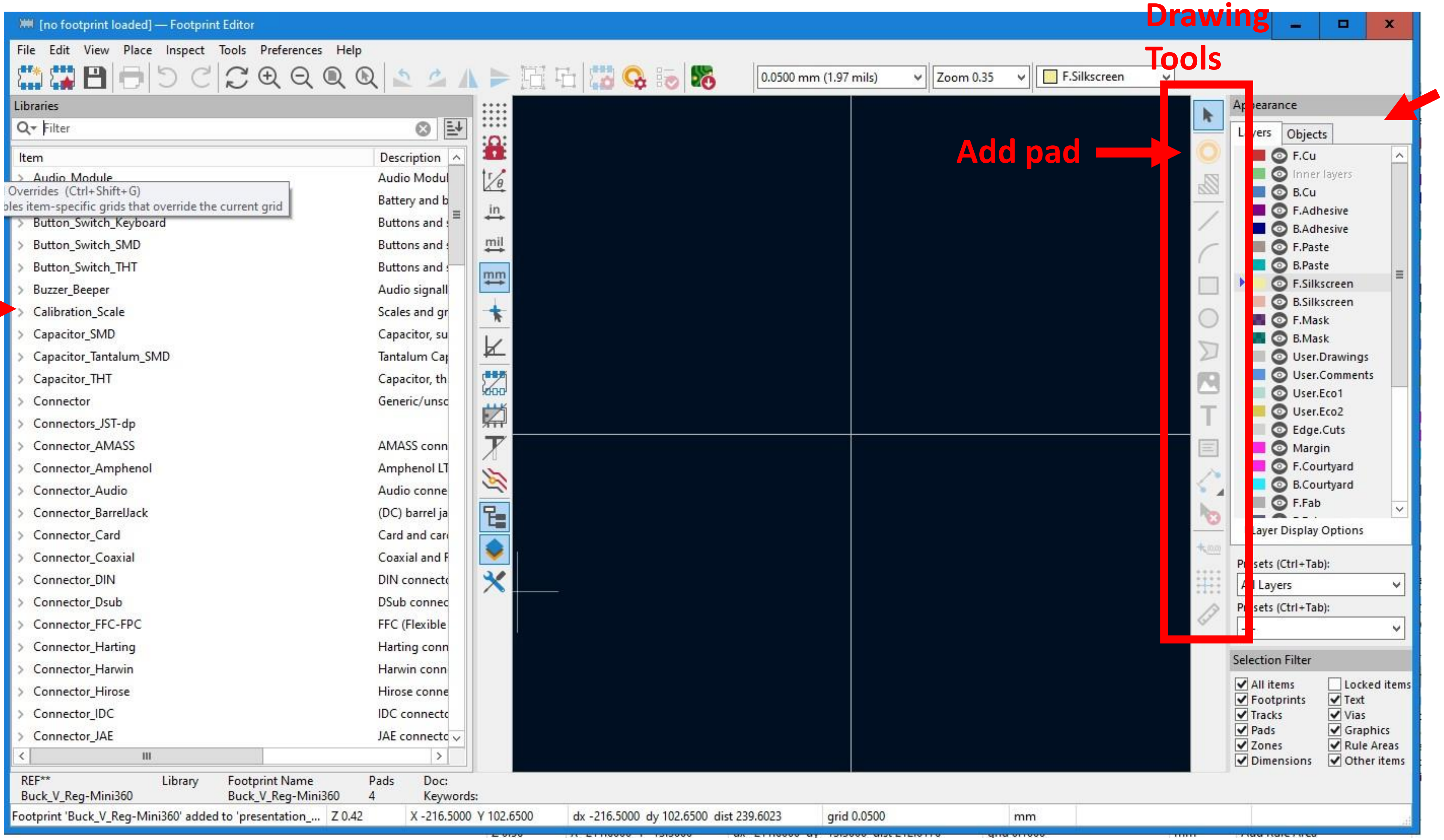
C1 through C6:

Capacitor_THT:C_Disc_D4.7mm_W2.5mm_P5.00mm

R1 through R4.

Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal

Footprint Editor



Drawing Tools

PCB Layers

Add pad

Existing Footprints

Overrides (Ctrl+Shift+G)

bles item-specific grids that override the current grid

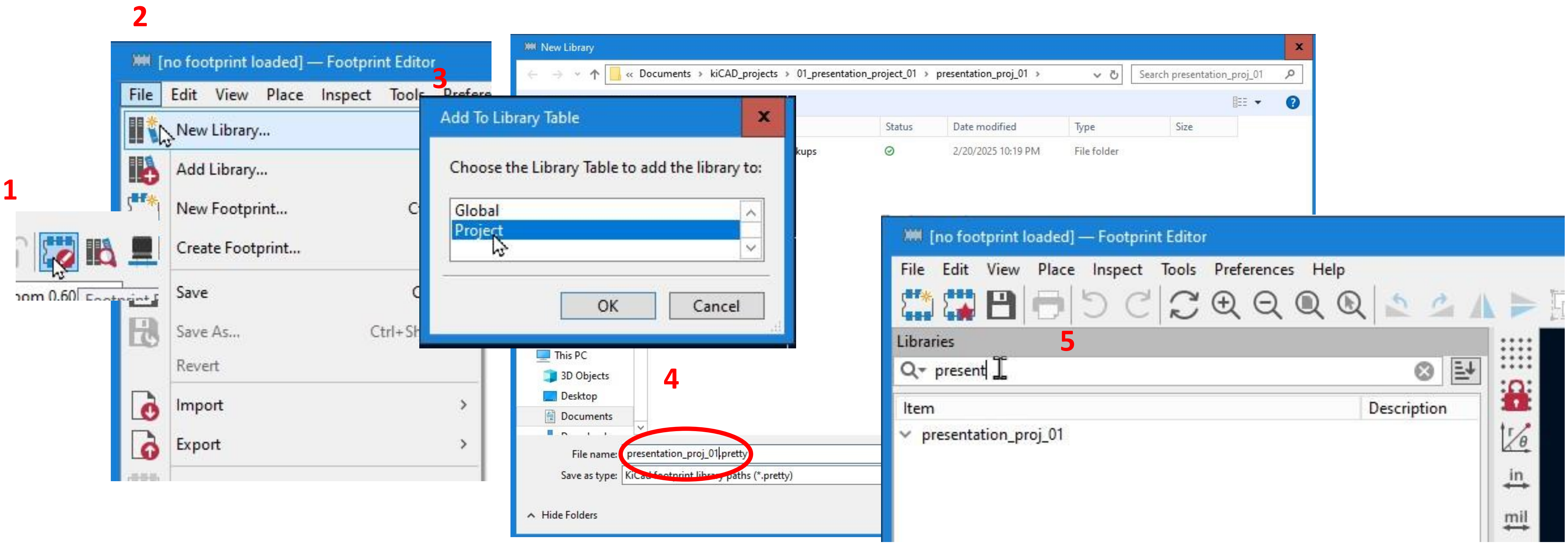
Item	Description
Audio_Module	Audio Modul
	Battery and b
Button_Switch_Keyboard	Buttons and
Button_Switch_SMD	Buttons and
Button_Switch_THT	Buttons and
Buzzer_Beeper	Audio signall
Calibration_Scale	Scales and gr
Capacitor_SMD	Capacitor, su
Capacitor_Tantalum_SMD	Tantalum Cap
Capacitor_THT	Capacitor, th
Connector	Generic/unsc
Connectors_JST-dp	
Connector_AMASS	AMASS conn
Connector_Amphenol	Amphenol LT
Connector_Audio	Audio conne
Connector_BarrelJack	(DC) barrel ja
Connector_Card	Card and car
Connector_Coaxial	Coaxial and F
Connector_DIN	DIN connecto
Connector_Dsub	DSub connec
Connector_FFC-FPC	FFC (Flexible
Connector_Harting	Harting conn
Connector_Harwin	Harwin conn
Connector_Hirose	Hirose conne
Connector_IDC	IDC connecto
Connector_JAE	JAE connecto

☑ All items ☐ Locked items
☑ Footprints ☑ Text
☑ Tracks ☑ Vias
☑ Pads ☑ Graphics
☑ Zones ☑ Rule Areas
☑ Dimensions ☑ Other items

REF**	Library	Footprint Name	Pads	Doc:
Buck_V_Reg-Mini360		Buck_V_Reg-Mini360	4	Keywords:

Footprint 'Buck_V_Reg-Mini360' added to 'presentation_... Z 0.42 X -216.5000 Y 102.6500 dx -216.5000 dy 102.6500 dist 239.6023 grid 0.0500 mm

Creating a Project Footprint Library


2
1
3
4
5
[no footprint loaded] — Footprint Editor
File Edit View Place Inspect Tools Prefere
New Library...
Add Library...
New Footprint...
Create Footprint...
Save
Save As... Ctrl+Sh
Revert
Import
Export
New Library
Documents › kiCAD_projects › 01_presentation_project_01 › presentation_proj_01 ›
Search presentation_proj_01
Status Date modified Type Size
2/20/2025 10:19 PM File folder
This PC
3D Objects
Desktop
Documents
Hide Folders
Add To Library Table
Global
Project
OK
Cancel
[no footprint loaded] — Footprint Editor
File Edit View Place Inspect Tools Preferences Help
Libraries
Item Description
presentation_proj_01

Choose the Library Table to add the library to:

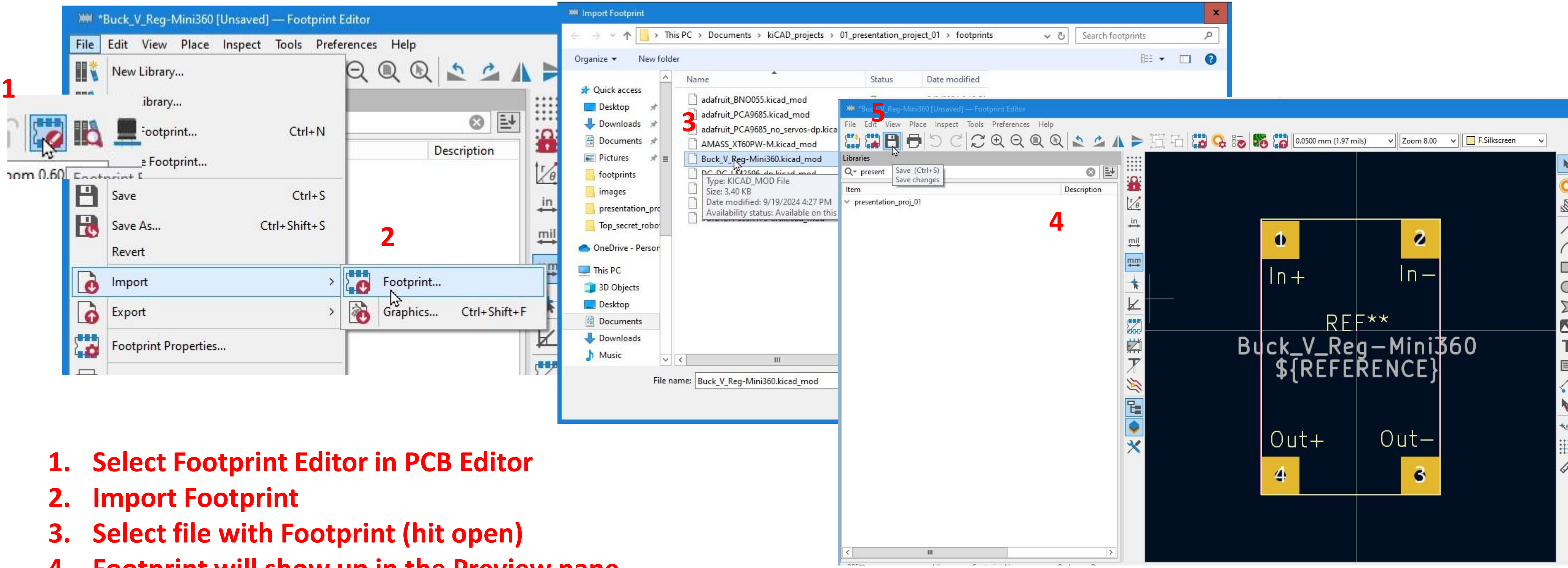
File name: presentation_proj_01.pretty

Save as type: KiCad footprint library paths (*.pretty)

present

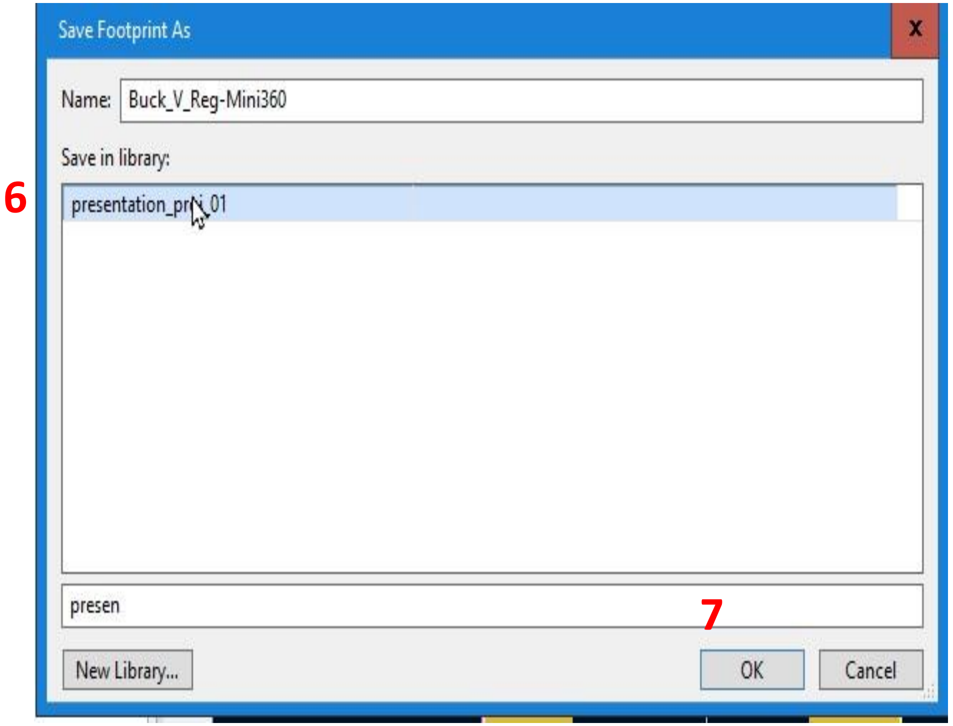
1. Select Footprint Editor in PCB Editor
2. Make a new library
3. Make the library a project library
4. Name and Save the library in your project
5. Search for the library to make sure it is created. It will be empty.

Adding a Footprint to Your Library



1. Select Footprint Editor in PCB Editor
2. Import Footprint
3. Select file with Footprint (hit open)
4. Footprint will show up in the Preview pane
5. Save

Adding a Footprint to Your Library



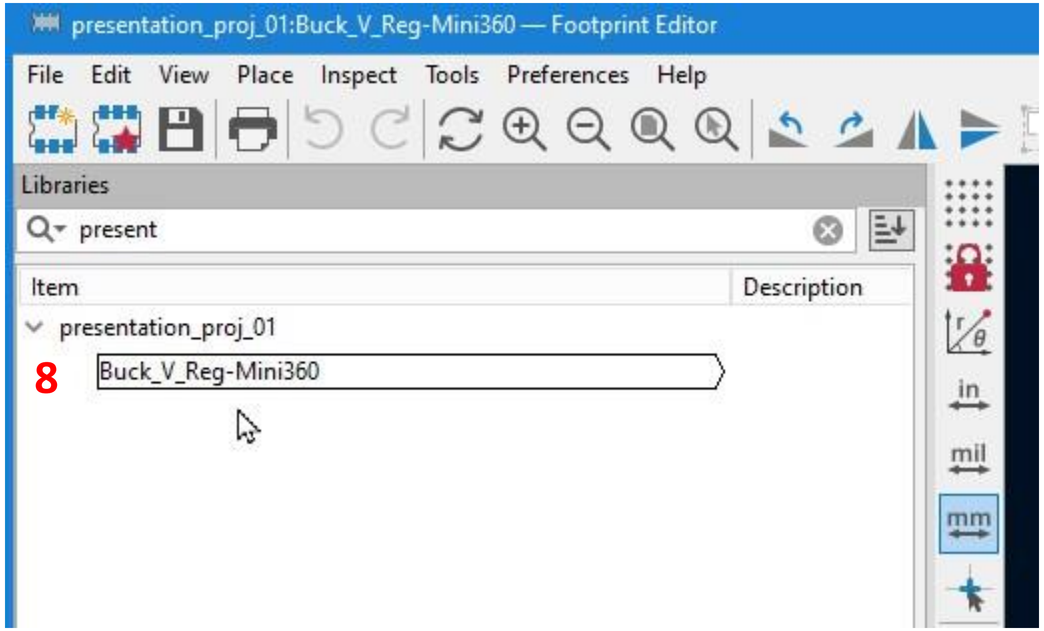
Save Footprint As

Name: Buck_V_Reg-Mini360

Save in library:

presentation_proj_01

presen



presentation_proj_01:Buck_V_Reg-Mini360 — Footprint Editor

Libraries: present

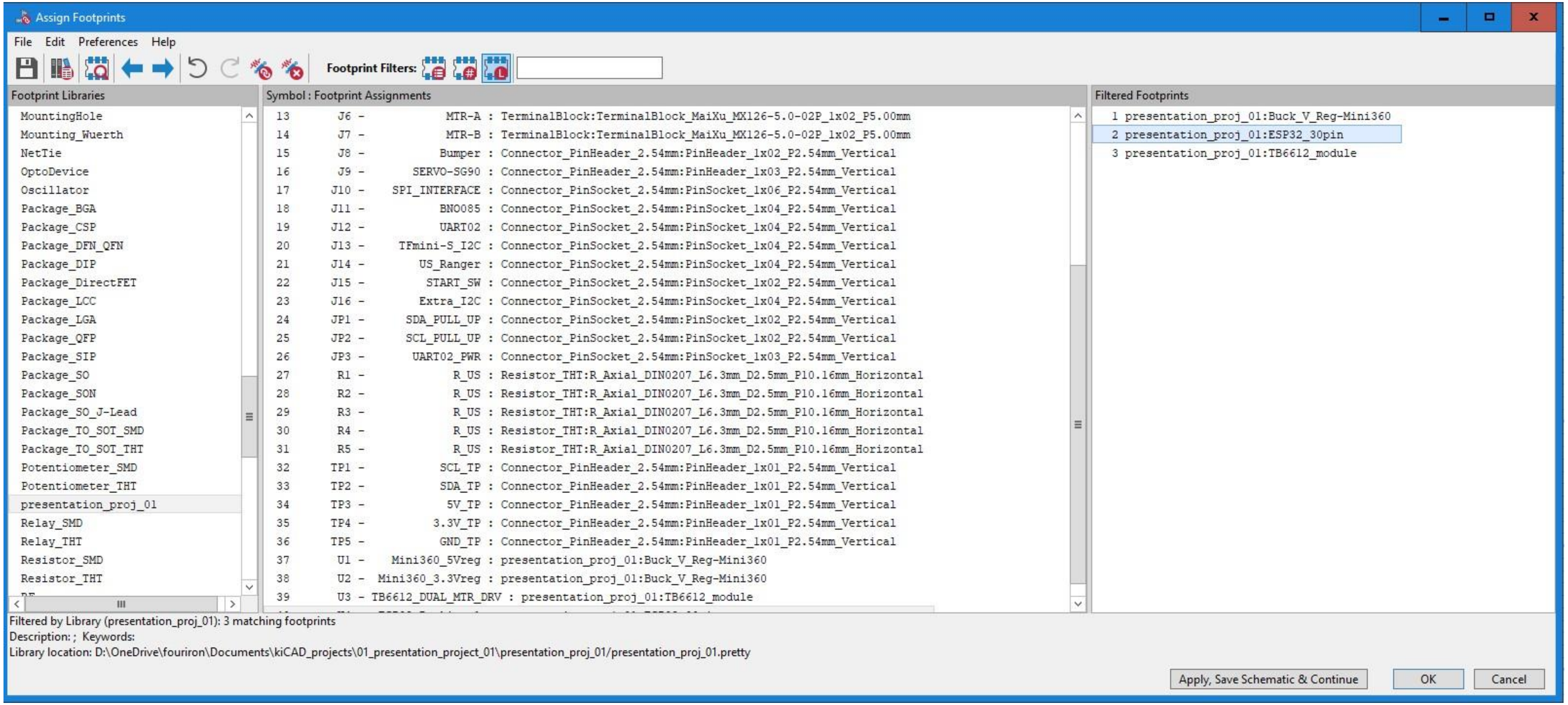
Item	Description
presentation_proj_01	
Buck_V_Reg-Mini360	

6. **Search for and select library**
7. **Click Ok**
8. **Check that Footprint shows up in library**

The ESP32 module had pin labeling issues.

ESP32_devkit_v1 sym and ftprint: https://forum.kicad.info/t/esp32-dev-ch340-c-symbol-and-footprint/56483/13
TB6612 module ftprint : https://www.snapeda.com/parts/ROB-14450/SparkFun/view-part/?welcome=home

All the Footprints Added


Assign Footprints
File Edit Preferences Help
Footprint Filters:
Footprint Libraries
Symbol : Footprint Assignments
Filtered Footprints
Apply, Save Schematic & Continue
OK
Cancel

Footprint Libraries

```
MountingHole
Mounting_Wuerth
NetTie
OptoDevice
Oscillator
Package_BGA
Package_CSP
Package_DFN_QFN
Package_DIP
Package_DirectFET
Package_LCC
Package_LGA
Package_QFP
Package_SIP
Package_SO
Package_SON
Package_SO_J-Lead
Package_TO_SOT_SMD
Package_TO_SOT_THT
Potentiometer_SMD
Potentiometer_THT
presentation_proj_01
Relay_SMD
Relay_THT
Resistor_SMD
Resistor_THT
```

Symbol : Footprint Assignments

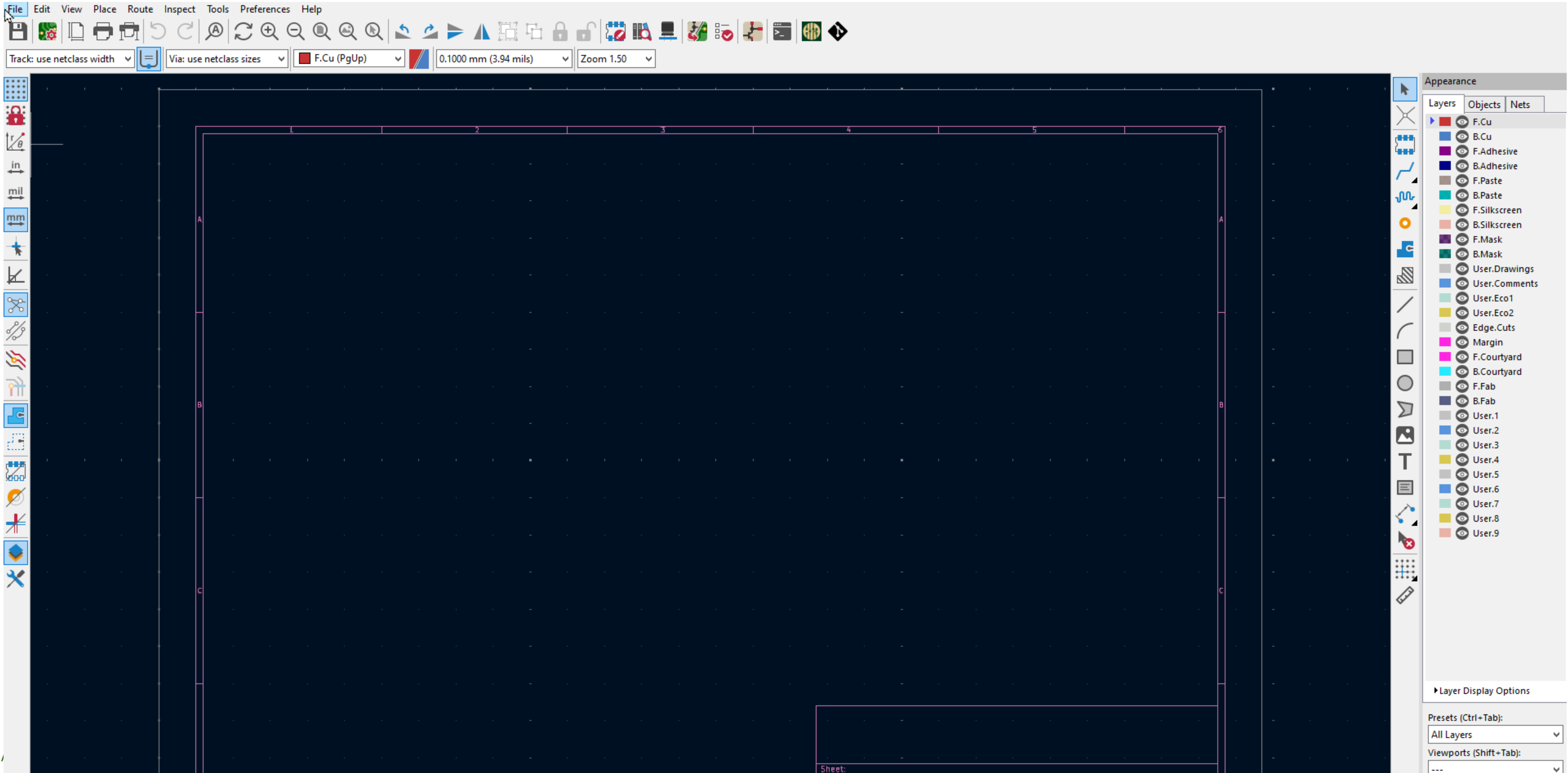
```
13      J6 -           MTR-A : TerminalBlock:TerminalBlock_MaiXu_MX126-5.0-02P_1x02_P5.00mm
14      J7 -           MTR-B : TerminalBlock:TerminalBlock_MaiXu_MX126-5.0-02P_1x02_P5.00mm
15      J8 -          Bumper : Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical
16      J9 -      SERVO-SG90 : Connector_PinHeader_2.54mm:PinHeader_1x03_P2.54mm_Vertical
17     J10 -   SPI_INTERFACE : Connector_PinSocket_2.54mm:PinSocket_1x06_P2.54mm_Vertical
18     J11 -          BNO085 : Connector_PinSocket_2.54mm:PinSocket_1x04_P2.54mm_Vertical
19     J12 -          UART02 : Connector_PinSocket_2.54mm:PinSocket_1x04_P2.54mm_Vertical
20     J13 -     TFmini-S_I2C : Connector_PinSocket_2.54mm:PinSocket_1x04_P2.54mm_Vertical
21     J14 -       US_Ranger : Connector_PinSocket_2.54mm:PinSocket_1x04_P2.54mm_Vertical
22     J15 -        START_SW : Connector_PinSocket_2.54mm:PinSocket_1x02_P2.54mm_Vertical
23     J16 -       Extra_I2C : Connector_PinSocket_2.54mm:PinSocket_1x04_P2.54mm_Vertical
24     JP1 -     SDA_PULL_UP : Connector_PinSocket_2.54mm:PinSocket_1x02_P2.54mm_Vertical
25     JP2 -     SCL_PULL_UP : Connector_PinSocket_2.54mm:PinSocket_1x02_P2.54mm_Vertical
26     JP3 -      UART02_PWR : Connector_PinSocket_2.54mm:PinSocket_1x03_P2.54mm_Vertical
27      R1 -            R_US : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
28      R2 -            R_US : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
29      R3 -            R_US : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
30      R4 -            R_US : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
31      R5 -            R_US : Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
32     TP1 -          SCL_TP : Connector_PinHeader_2.54mm:PinHeader_1x01_P2.54mm_Vertical
33     TP2 -          SDA_TP : Connector_PinHeader_2.54mm:PinHeader_1x01_P2.54mm_Vertical
34     TP3 -           5V_TP : Connector_PinHeader_2.54mm:PinHeader_1x01_P2.54mm_Vertical
35     TP4 -         3.3V_TP : Connector_PinHeader_2.54mm:PinHeader_1x01_P2.54mm_Vertical
36     TP5 -          GND_TP : Connector_PinHeader_2.54mm:PinHeader_1x01_P2.54mm_Vertical
37      U1 -     Mini360_5Vreg : presentation_proj_01:Buck_V_Reg-Mini360
38      U2 -   Mini360_3.3Vreg : presentation_proj_01:Buck_V_Reg-Mini360
39      U3 - TB6612_DUAL_MTR_DRV : presentation_proj_01:TB6612_module
```

Filtered Footprints

```
1 presentation_proj_01:Buck_V_Reg-Mini360
2 presentation_proj_01:ESP32_30pin
3 presentation_proj_01:TB6612_module
```

Filtered by Library (presentation_proj_01): 3 matching footprints
Description: ; Keywords:
Library location: D:\OneDrive\fouriron\Documents\kiCAD_projects\01_presentation_project_01\presentation_proj_01/presentation_proj_01.pretty

Finally, the PCB Editor



Track: use netclass width

Via: use netclass sizes

F.Cu (PgUp)

0.1000 mm (3.94 mils)

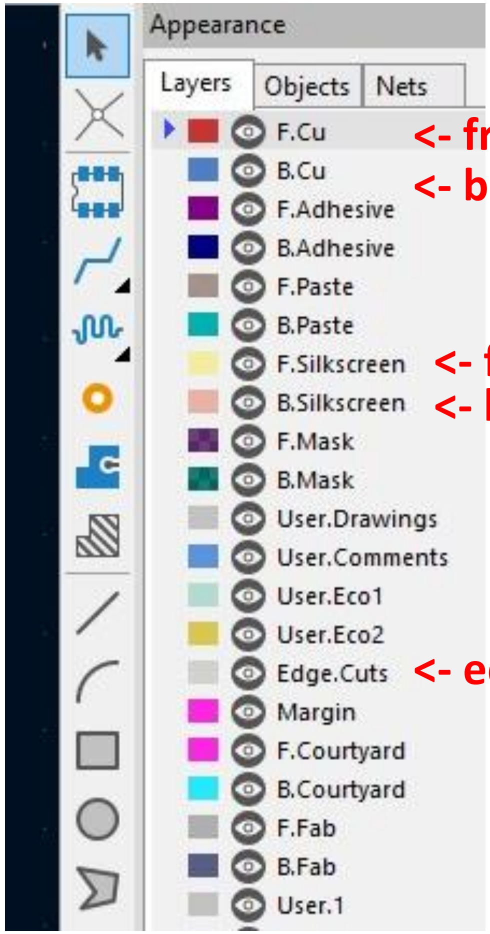
Zoom 1.50

Presets (Ctrl+Tab): All Layers

Viewports (Shift+Tab): ---

PCB Editor Tools

- The <u>right menu </u>has what you need to make a PCB.



Appearance

Layers | Objects | Nets

- 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

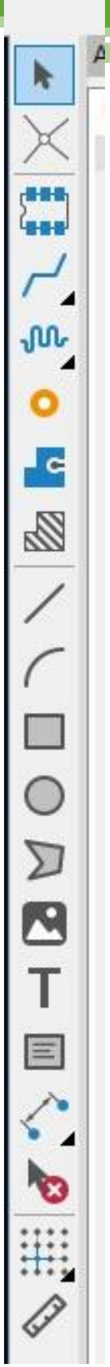
<- front copper

<- backside copper

<- front silkscreen

<- backside silkscreen

<- edge cuts (PCB board shape)



<- select item - IMPORTANT

<- local Ratsnest

<- add footprint - IMPORTANT

<- route tracks - IMPORTANT

<- tune length of single track

<- add vias - IMPORTANT

<- add filled zone - IMPORTANT

<- add Rule area

Drawing Tools tools

<- add text

<- add textbox

<- add dimensions

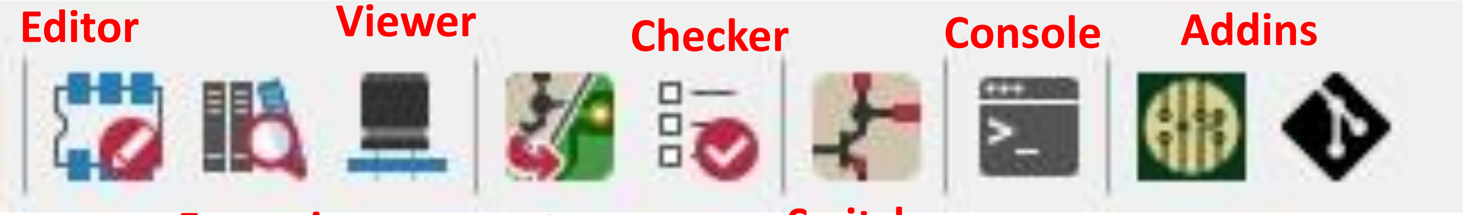
<- deletion tool

<- set grid origin or drill/place orgin

<- interactively measure

PCB Editor

- The top menu has tools that you will also need.



Footprint Editor

3D Viewer

Design Rule Checker

Scripting Console

Addins

Footprint Library

Update PCB from Schematic

Switch to Schematic Editor

The next step

Reflection on What We Have Done

- Completed the schematic, by making the design, adding/making the symbols, and completed the wiring.
- Assigned net classes to and passed ERC.
- Assigned footprints to all the schematic's parts.

REMAINING:

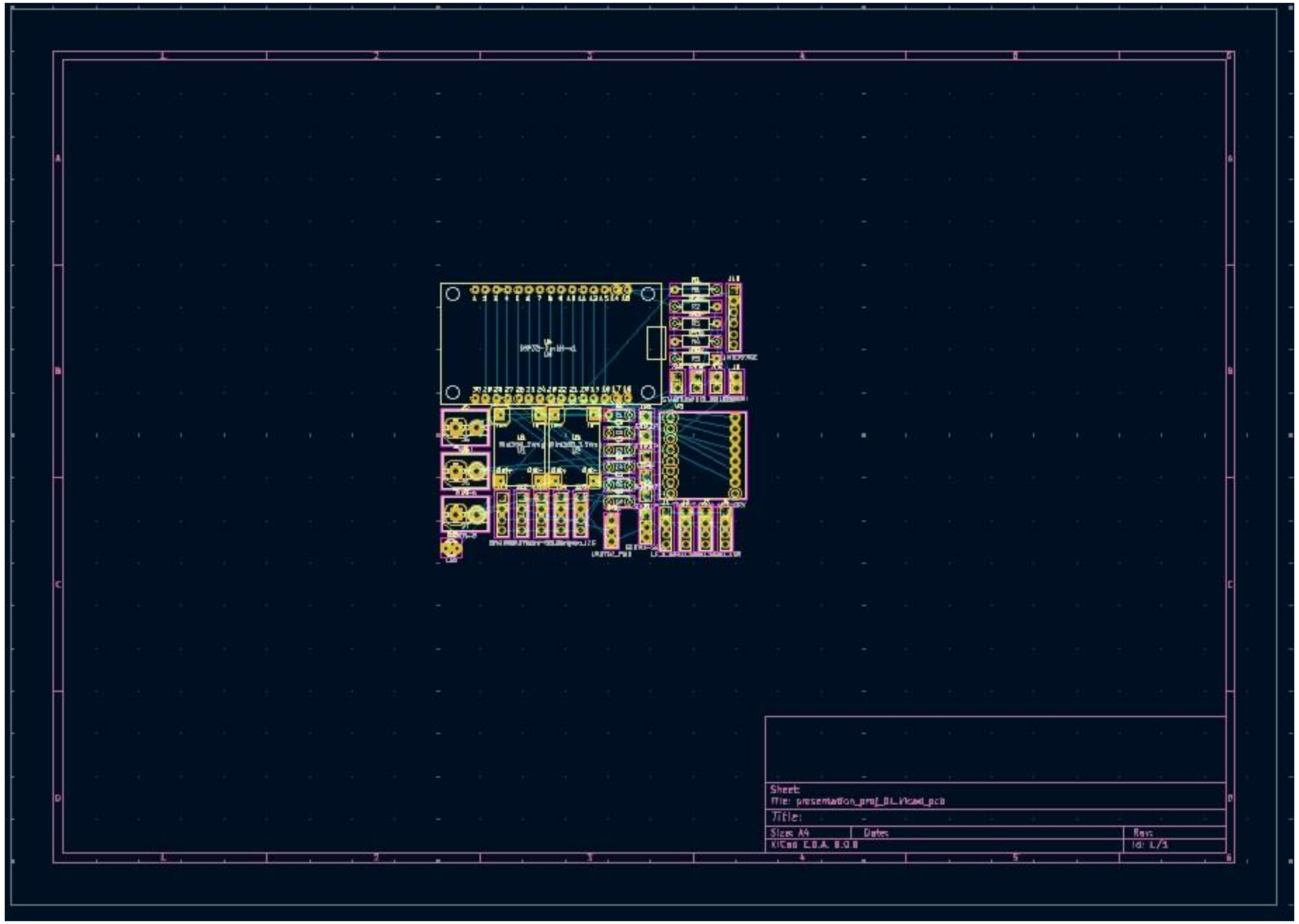
- Placing the footprints and the ratsnest on the PCB and arranging
  the footprints in a logical order.
- Route the wiring. We will use an auto-router.
- Print a copy of the final PCB and hand place the parts on it to make sure that everything looks good.
- Send the finished PCB design to a PCB manufacturer.

Updating the PCB from the Schematic

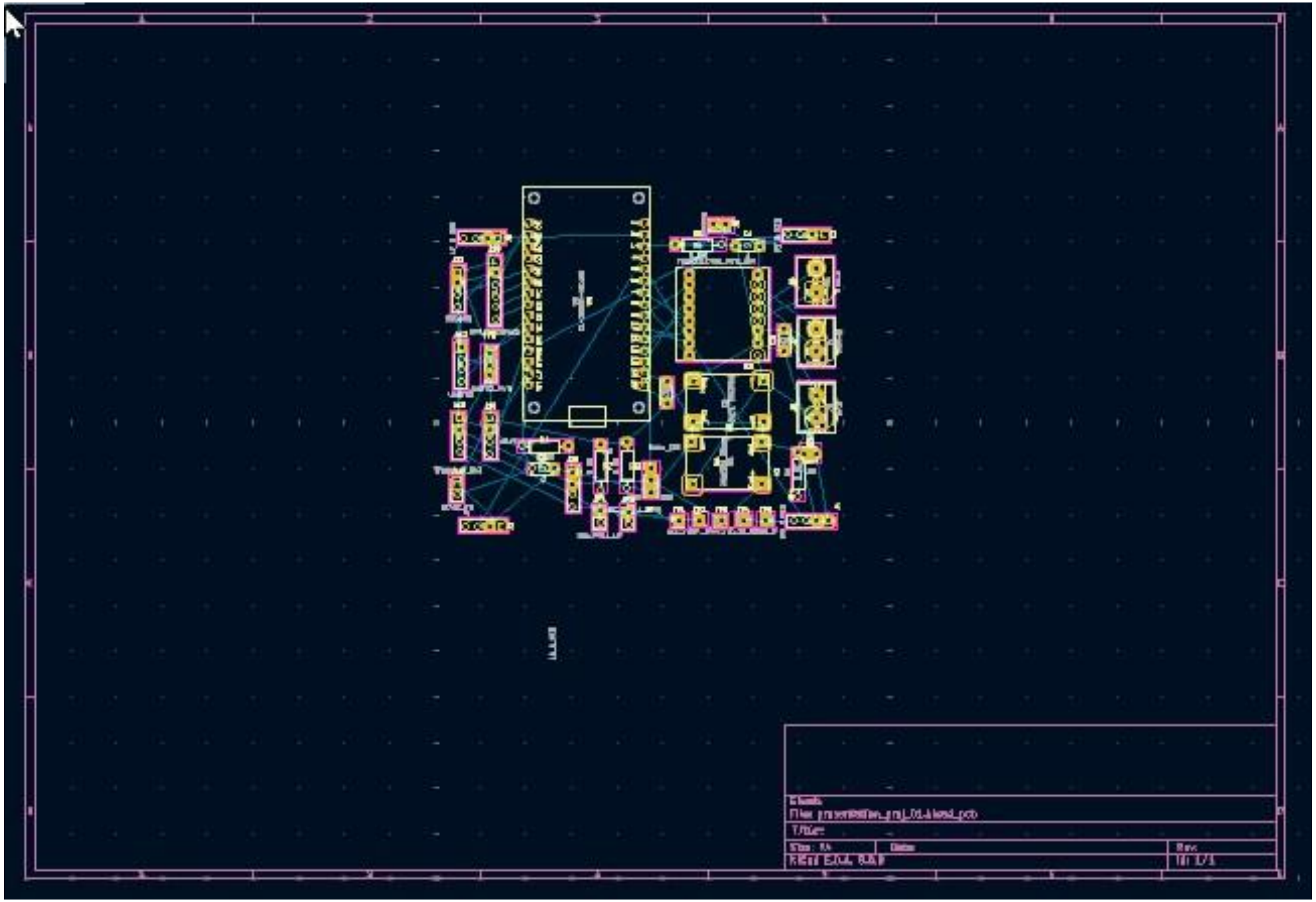
This places all the part on the PC Editor and connects them with the “ratsnest” which represents the wiring.

Next step is to rearrange the footprints:

- There should be a drop sensor in each corner.
- The antenna of the ESP32 needs to hang over the edge of the board.
- The distance sensors should be near the front of the board.
- The motor driver should be on the edge to connect to the terminal blocks. Same with the battery terminal block and voltage regulators.



First Shot


Sheet:
File: presentation_proj_01.kicad_pcb
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. 8.0.8
Id: 1/1

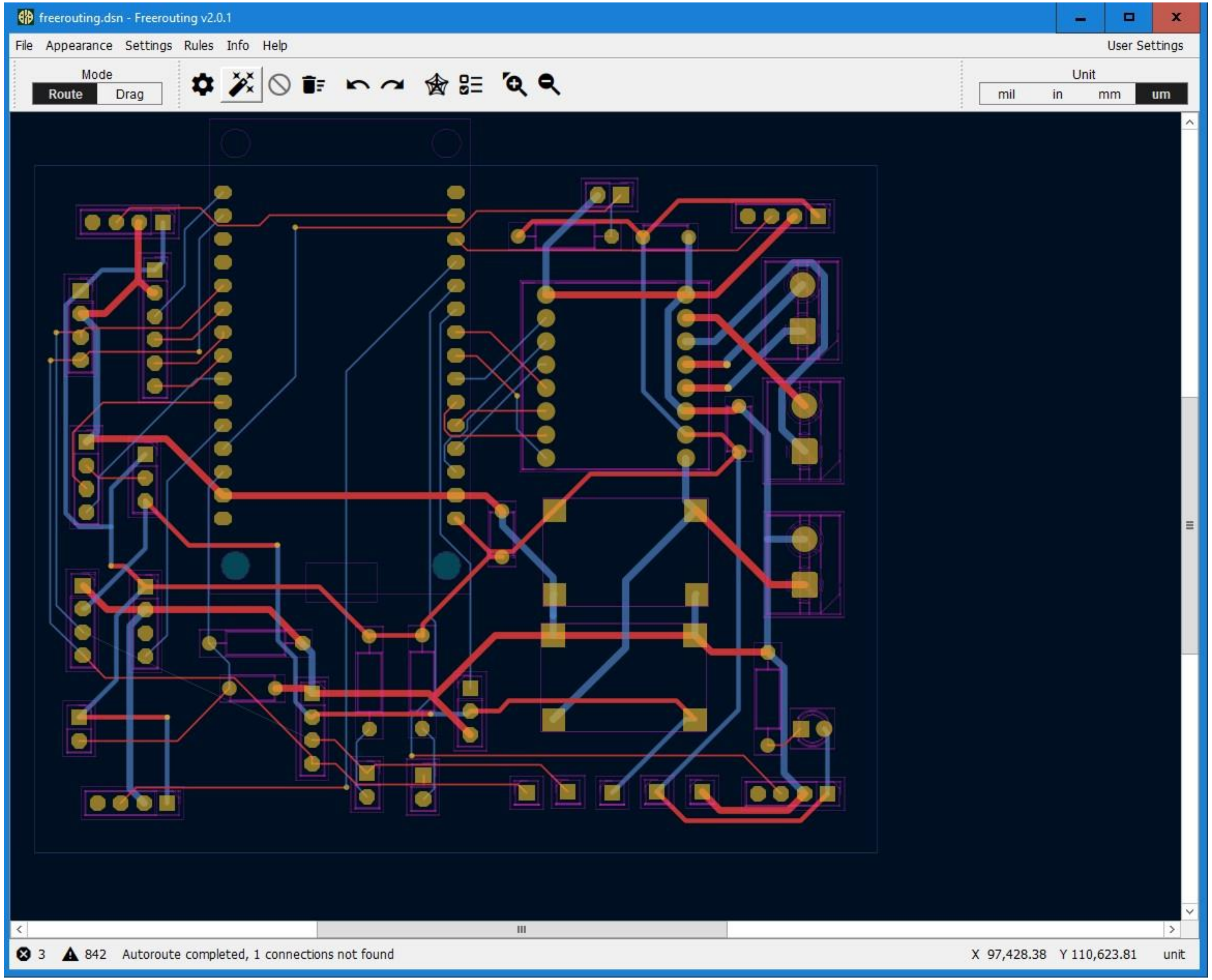
Auto Routing

Freerouting →

Freerouting is a very nice and fast auto-router.

I prefer to unclick the "restrict pin exit directions" In the Routing Settings.

You need to add a board Border before running Freerouting.



freerouting.dsn - Freerouting v2.0.1

3 842 Autoroute completed, 1 connections not found

X 97,428.38 Y 110,623.81 unit

Board boundaries

- KiCad is not very good in detailed graphics, and when board outlines become really complex, they should be drawn in an external program and imported into KiCad.

- **I prefer to use Inkscape to draw my board boundaries. Some resources:**

KiCad - Complex Edge Cuts in KiCAD 5.x with Inkscape
https://www.youtube.com/watch?v=wzvZ9Ssn0eo
note: New versions KiCad can import .svg (i.e., native Inkscape) files and don't use .dxf files.

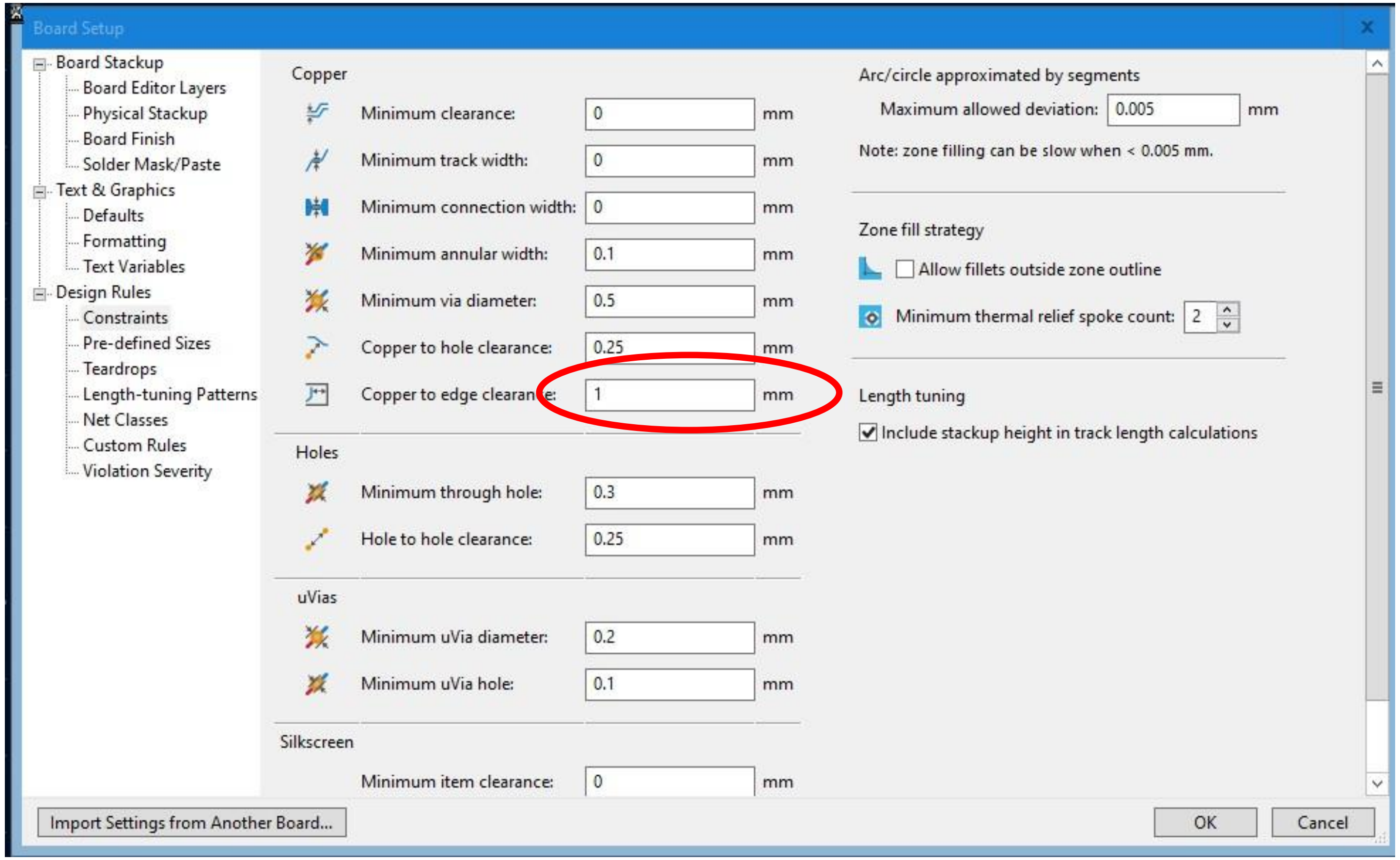
KiCad 6 - Importing Complex Board Outlines as Vector Graphics
https://community.element14.com/members-area/b/blog/posts/kicad-6---importing-complex-board-outlines-as-vector-graphics

KiCAD - Creating a PCB to exact dimensions
https://www.youtube.com/watch?v=wzvZ9Ssn0eo&t=8s

Fill Zones

Using File>Board Setup in the PCB Editor, select “Copper to edge clearance” under Design Rules >Constraints.

Change value to 1.0mm.



Copper

Minimum clearance: 0 mm

Minimum track width: 0 mm

Minimum connection width: 0 mm

Minimum annular width: 0.1 mm

Minimum via diameter: 0.5 mm

Copper to hole clearance: 0.25 mm

Copper to edge clearance: 1 mm

Holes

Minimum through hole: 0.3 mm

Hole to hole clearance: 0.25 mm

uVias

Minimum uVia diameter: 0.2 mm

Minimum uVia hole: 0.1 mm

Silkscreen

Minimum item clearance: 0 mm

Arc/circle approximated by segments

Maximum allowed deviation: 0.005 mm

Note: zone filling can be slow when < 0.005 mm.

Zone fill strategy

☐ Allow fillets outside zone outline

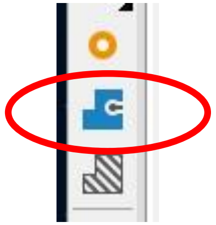
Minimum thermal relief spoke count: 2

Length tuning

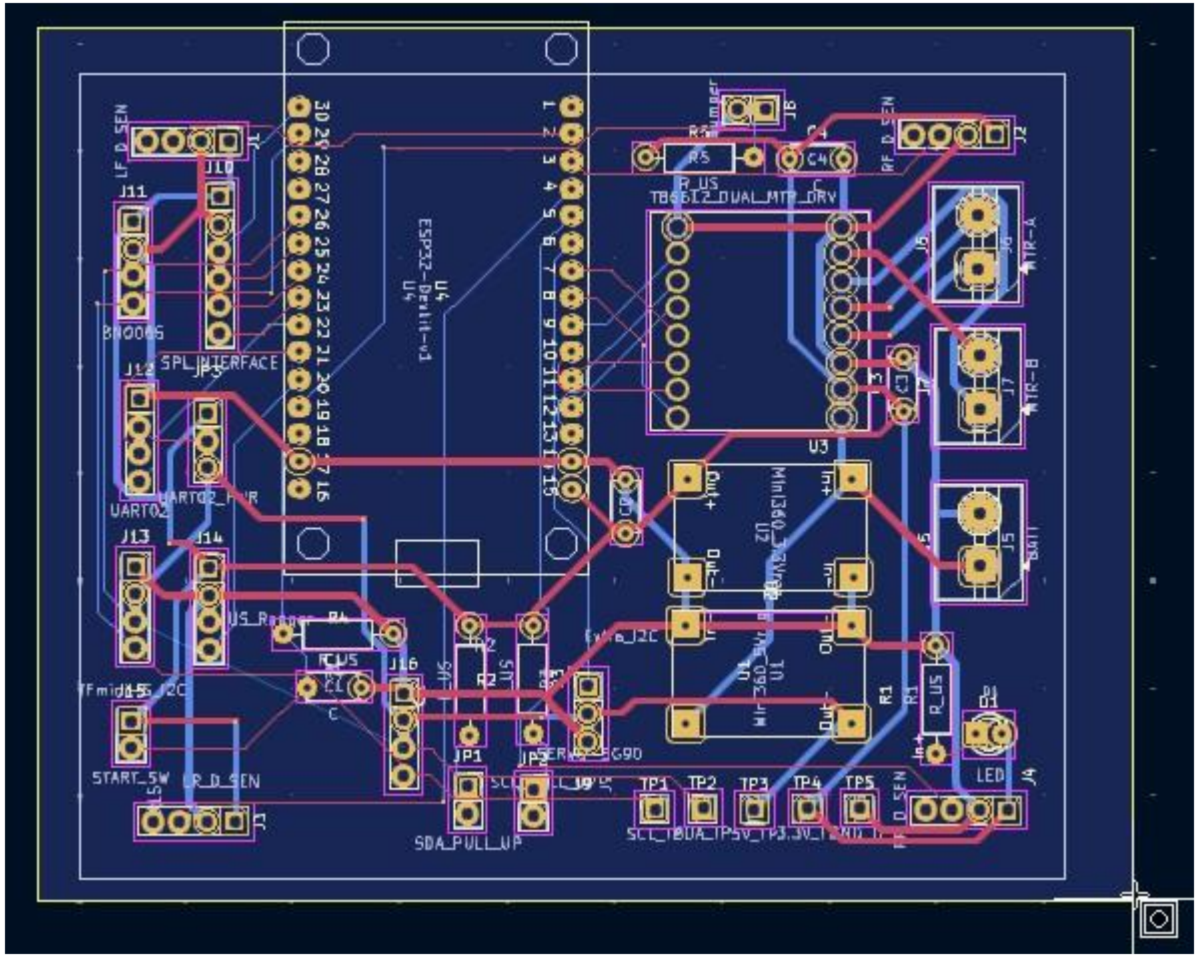
☑ Include stackup height in track length calculations

Fill Zones

Fill Zone Tool

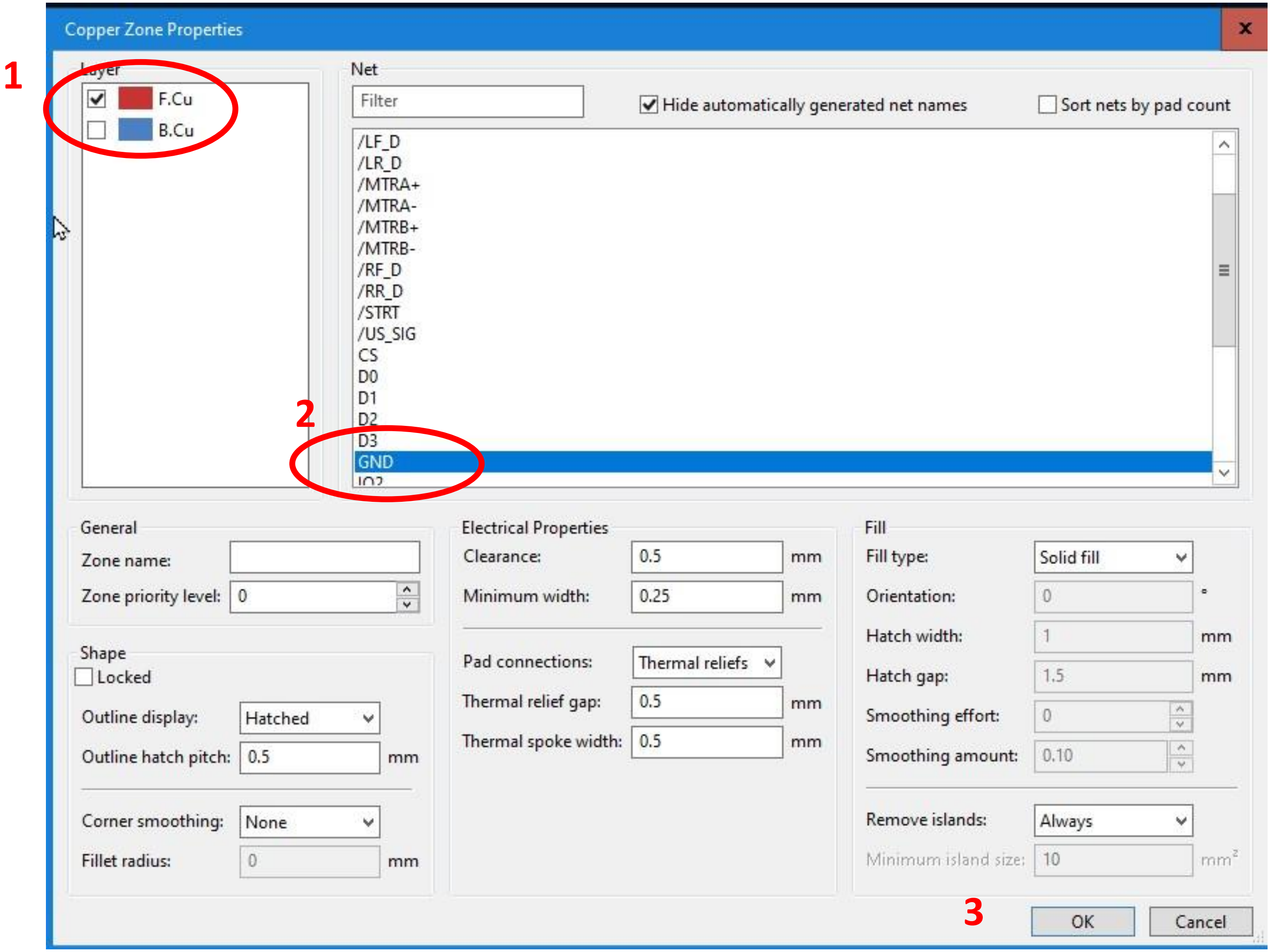


Draw zone around the entire board.



Fill Zones

1. Select a Cu Layer.
2. Select a net to fill the zone.
3. Click OK.



Layer
- ☑ 🟥 F.Cu
- □ 🟦 B.Cu

Net

Filter

☑ Hide automatically generated net names

□ Sort nets by pad count

/LF_D
/LR_D
/MTRA+
/MTRA-
/MTRB+
/MTRB-
/RF_D
/RR_D
/STRT
/US_SIG
CS
D0
D1
D2
D3
GND
IO2

General

Zone name:

Zone priority level: 0

Shape

□ Locked

Outline display: Hatched

Outline hatch pitch: 0.5 mm

Corner smoothing: None

Fillet radius: 0 mm

Electrical Properties

Clearance: 0.5 mm

Minimum width: 0.25 mm

Pad connections: Thermal reliefs

Thermal relief gap: 0.5 mm

Thermal spoke width: 0.5 mm

Fill

Fill type: Solid fill

Orientation: 0 °

Hatch width: 1 mm

Hatch gap: 1.5 mm

Smoothing effort: 0

Smoothing amount: 0.10

Remove islands: Always

Minimum island size: 10 mm^2

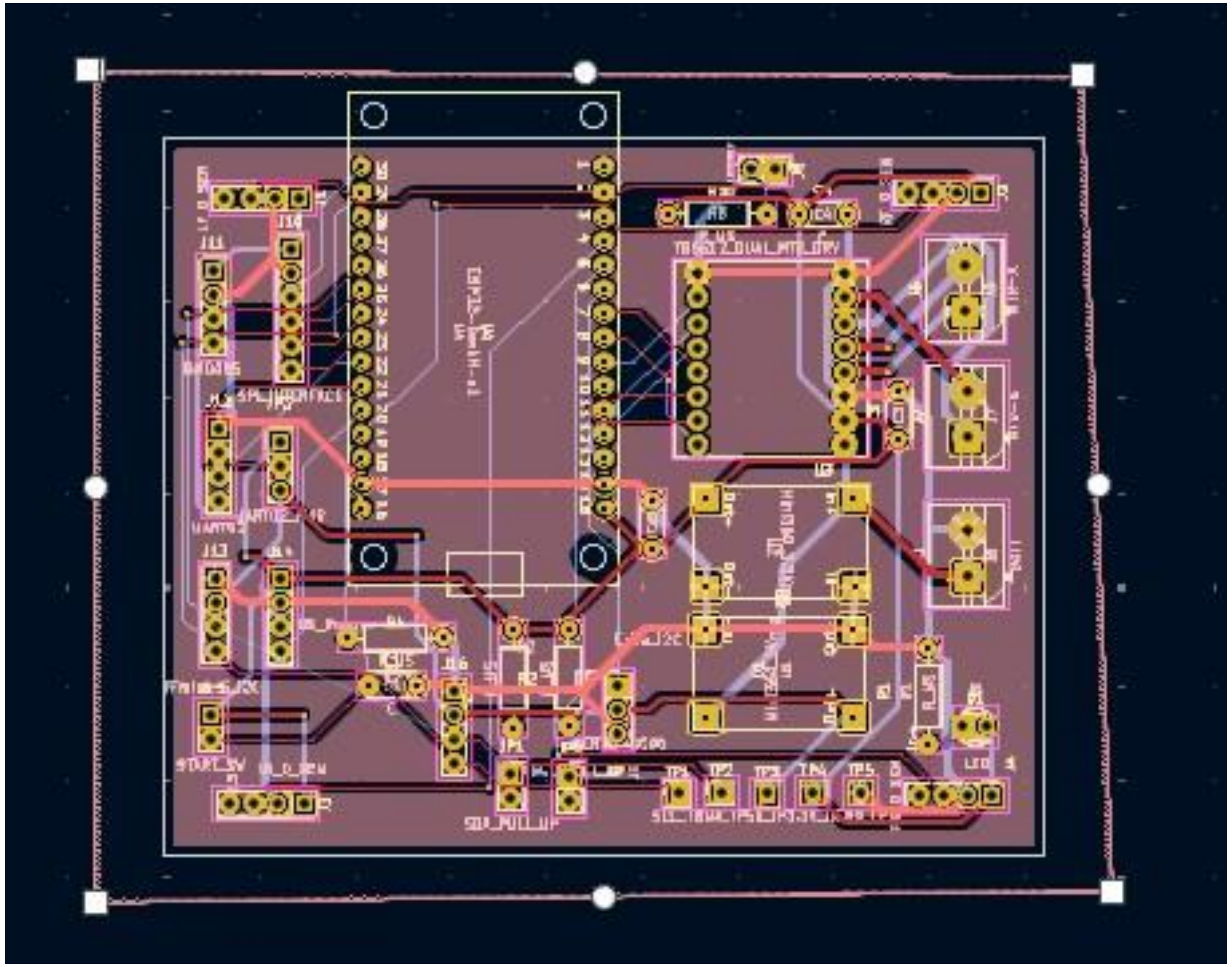
Fill Zones

Press “b” to refill zones.

Repeat for backside Cu.

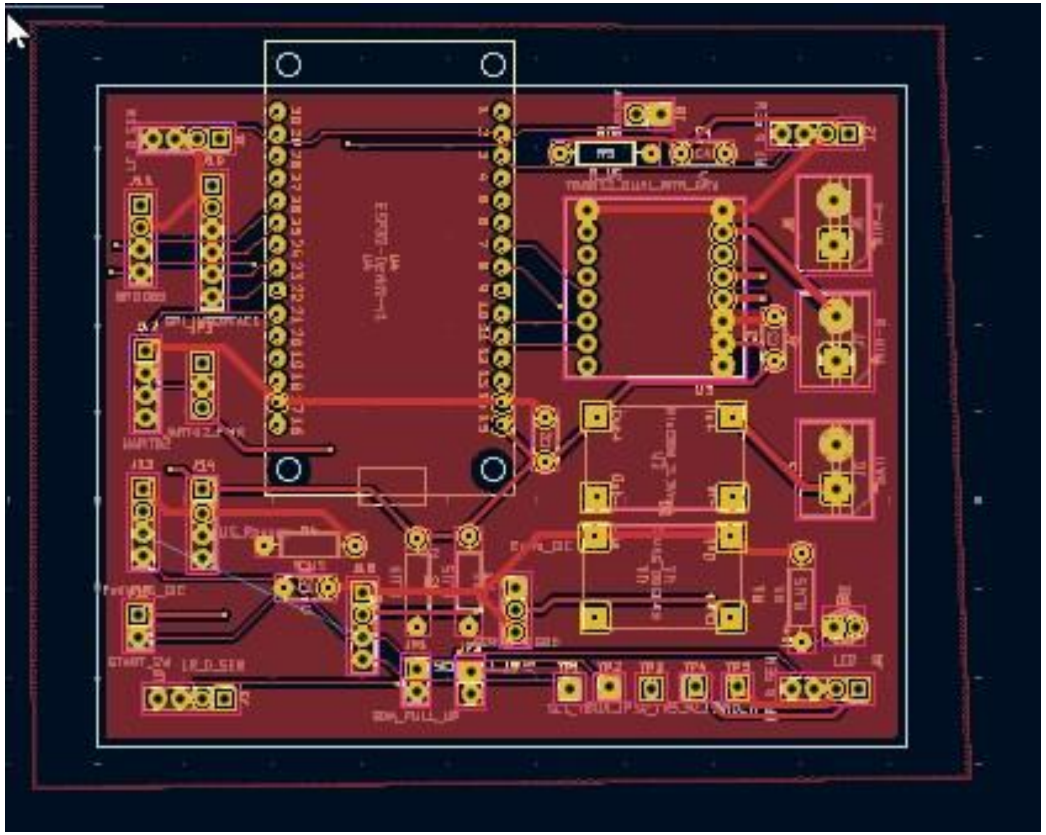
Use net 3.3V.

Use ctrl b to unfill.

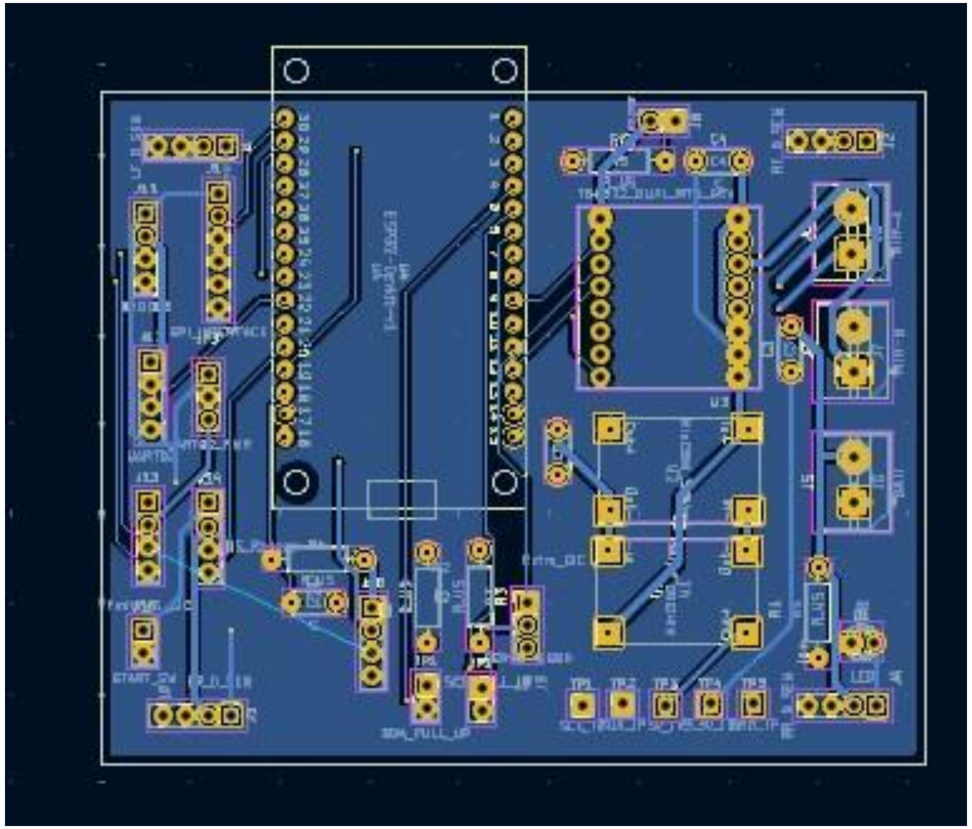


Fill Zones – Completed

Front Cu Layer



Backside Cu Layer



Run ERC – before making Gerber files

ERC will tell what errors you have in your PCB placements.

Generally, the errors are can be easily fixed.

Look at the warnings, but usually they can be ignored.

Producing the Gerber Files

For JLCPCB use the instructions from this link: https://jlcpcb.com/help/article/how-to-generate-gerber-and-drill-files-in-kicad-8

After making the Gerber, drill, and drill map files, it is a good idea to use a 3rd party Gerber viewer. JLCPCB recommends their own JLCPCB Gerber Viewer or open-source Gerbv.

JLCPCB Gerber Viewer: https://jlcpcb.com/RGE

Gerbv download: https://gerbv.github.io/

The End

All Robots Start with An Idea – A Tabletop Robot

Parts:

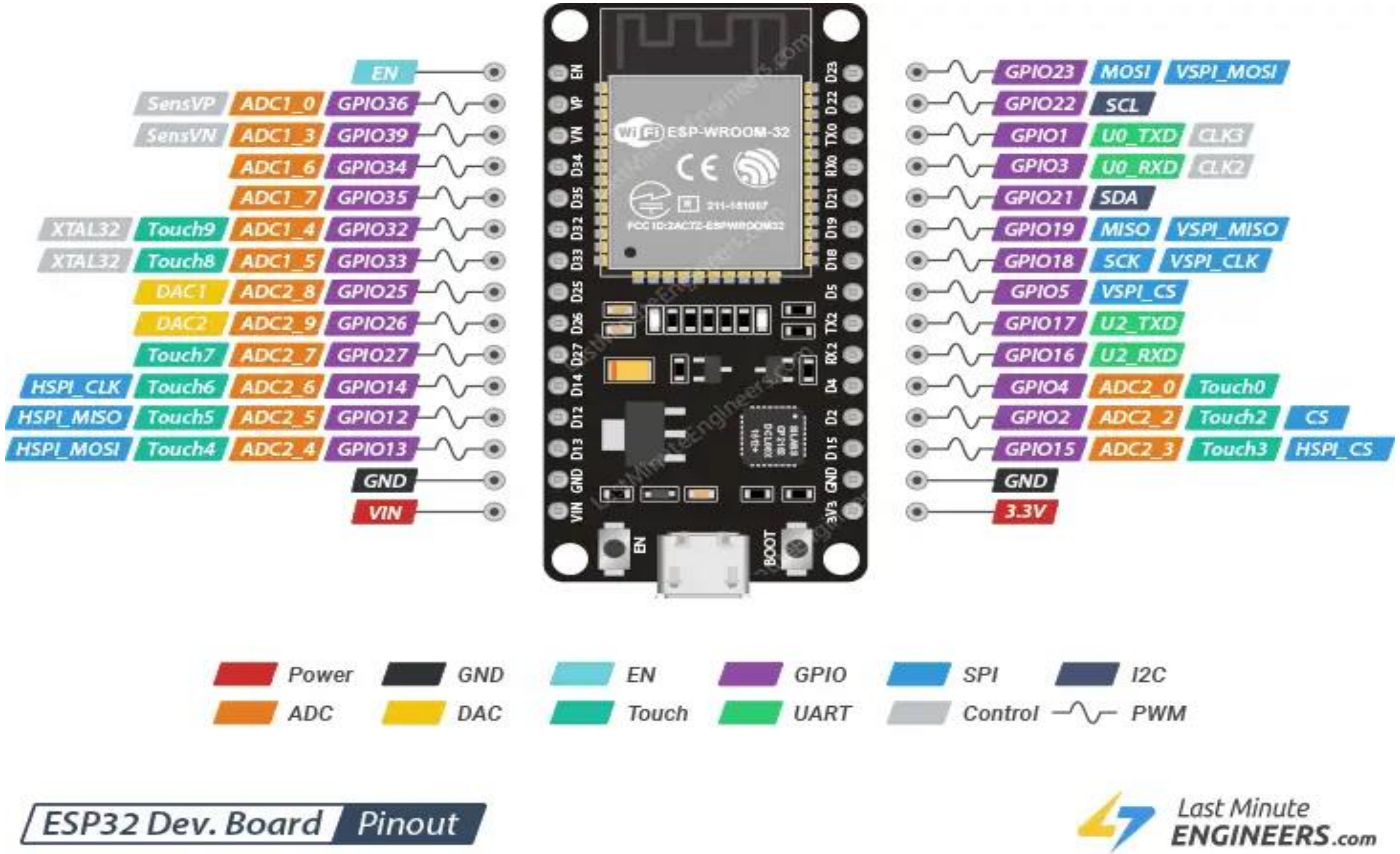
- 2 yellow TTmotor motors from DFRobot with quadrature encoders
- ESP32-WROOM-32 devkit v1 (30 pin version) module ✓
- 4 IR drop sensors – HiLetgo TCRT5000 IR sensors (GPIO)
- Dual H bridge for motors – ZK-5AD module (GPIO) or TB6612FNG (GPIO) ✓
- 9 DOF IMU – BNO085 (I2C/UART)
- 7.2V to 5V regulator - 5V version of Mini360 buck converter (Pwr) ✓
- 7.2V to 3.3V regulator – 3.3V version of Mini360 buck converter (Pwr) ✓
- Distance sensor – either a HSR04 ultrasonic (GPIO) or a TFMini-S ToF LiDAR (UART/I2C)
- Power and Start switches, “on” LED with current limiting resistor
- Bumper (optional – should have provisions for)

✓ = on PCB - other parts will have headers on PCB.

- P

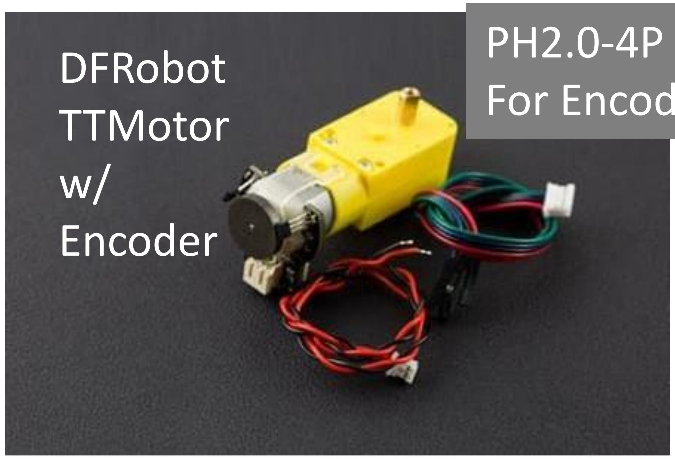
Parts - ESP32 DevKit v1

2-2.54mm-15p headers



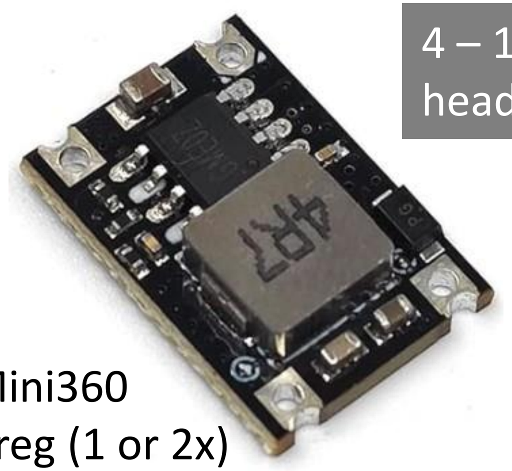
There is an additional UART – you have to assign the pins.

Parts - other



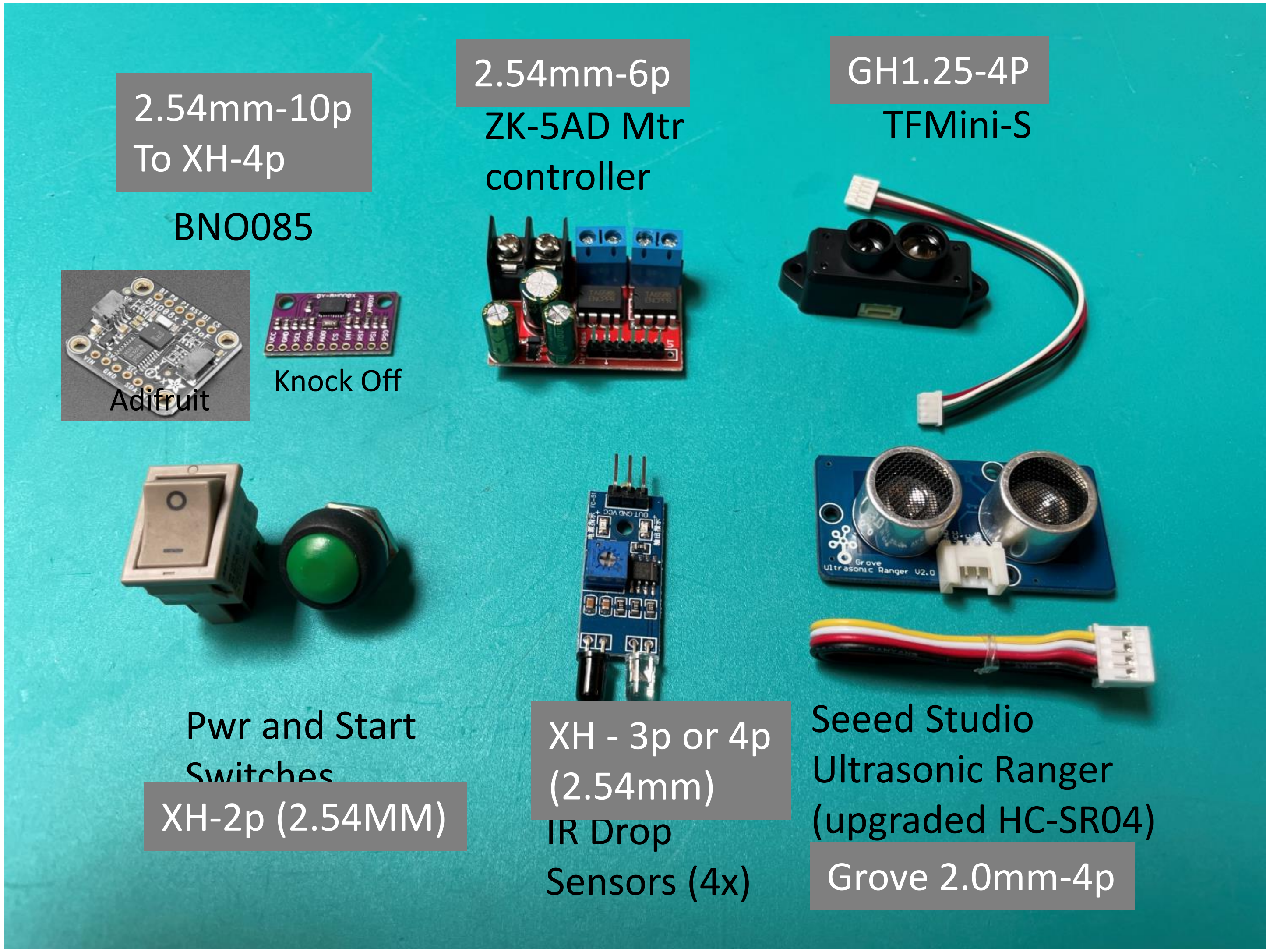
DFRobot
TTMotor
w/
Encoder

PH2.0-4P
For Encoders



4 – 1p
headers

Mini360
Vreg (1 or 2x)



2.54mm-10p
To XH-4p

BNO085

Adifruit

Knock Off

2.54mm-6p
ZK-5AD Mtr
controller

GH1.25-4P
TFMini-S

Pwr and Start
Switches

XH-2p (2.54MM)

XH - 3p or 4p
(2.54mm)

IR Drop
Sensors (4x)

Seeed Studio
Ultrasonic Ranger
(upgraded HC-SR04)

Grove 2.0mm-4p

Ideally, we will have polarized headers for all the parts on our PCB.