



PCB Design with the KiCad EDA

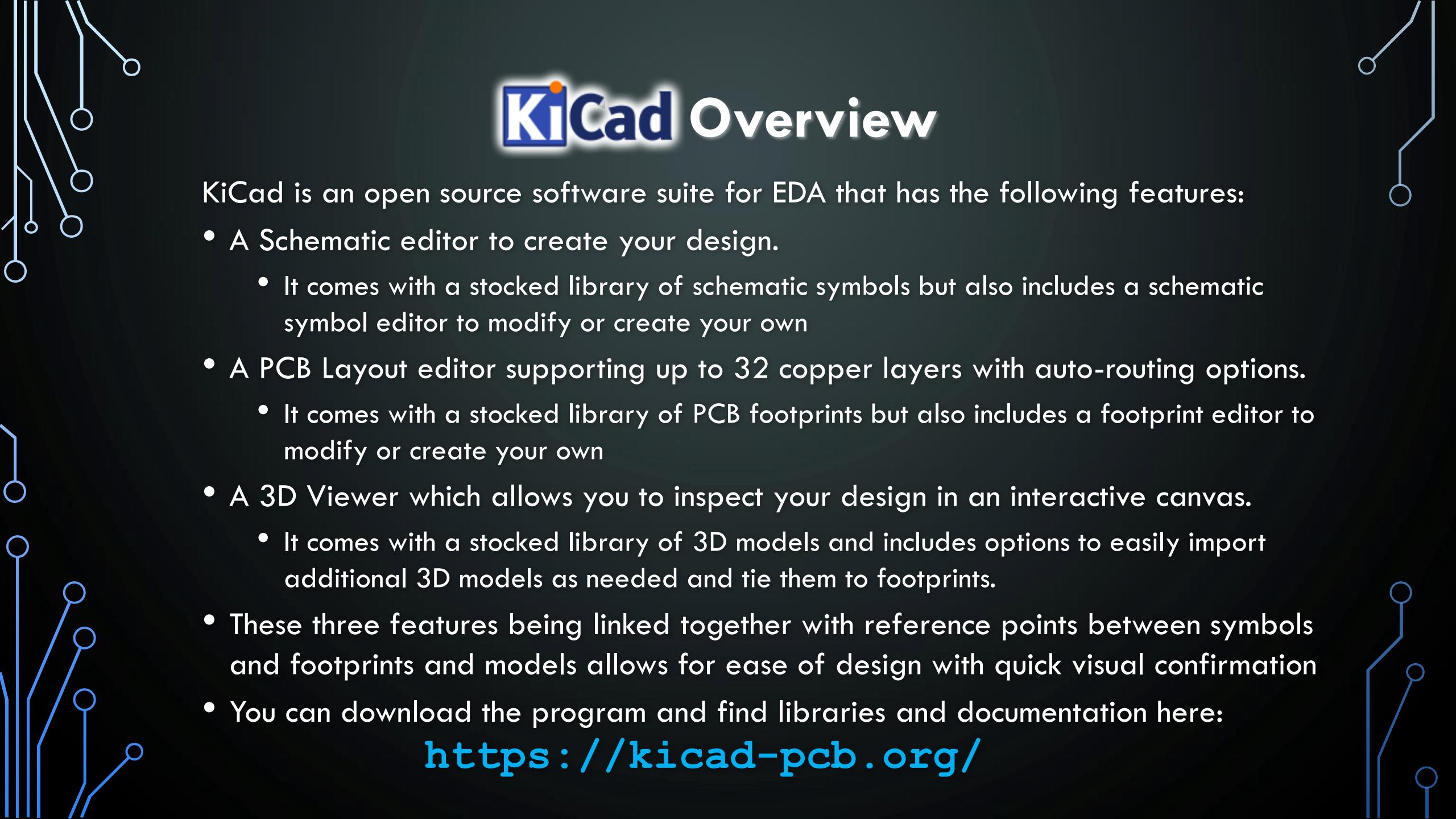
PRESENTED BY RICHARD GOWEN (@alt_bier)

Created for BSidesDFW 2020 HHV

This Slide Deck Is Available at <https://altbier.us/pcb/>

What are all these acronyms?

- PCB = Printed Circuit Board
 - A PCB mechanically supports and electrically connects electronic components using conductive tracks and pads that are etched from copper laminated on sheets of a non-conductive substrate. The pattern is etched using a form of photolithography allowing mass duplication like photographs from a film negative thus ‘printing’ the circuit boards.
- EDA = Electronic Design Automation
 - A category of software tools for designing electronic systems such as integrated circuits and printed circuit boards.
- KiCad = “Ki” + Computer Aided Design
 - The term ‘Ki’ (pronounced Key even though I keep saying it wrong) is a reference to a company that the original software author Jean-Pierre Charras knew.
 - Computer Aided Design refers to the use of computers in the creation, modification, analysis, or optimization of a design.
 - KiCad is an open source software suite for electronic design automation.



KiCad Overview

KiCad is an open source software suite for EDA that has the following features:

- A Schematic editor to create your design.
 - It comes with a stocked library of schematic symbols but also includes a schematic symbol editor to modify or create your own
- A PCB Layout editor supporting up to 32 copper layers with auto-routing options.
 - It comes with a stocked library of PCB footprints but also includes a footprint editor to modify or create your own
- A 3D Viewer which allows you to inspect your design in an interactive canvas.
 - It comes with a stocked library of 3D models and includes options to easily import additional 3D models as needed and tie them to footprints.
- These three features being linked together with reference points between symbols and footprints and models allows for ease of design with quick visual confirmation
- You can download the program and find libraries and documentation here:

<https://kicad-pcb.org/>

Project File Layout

KiCad will create files in a project directory when you work with various features.

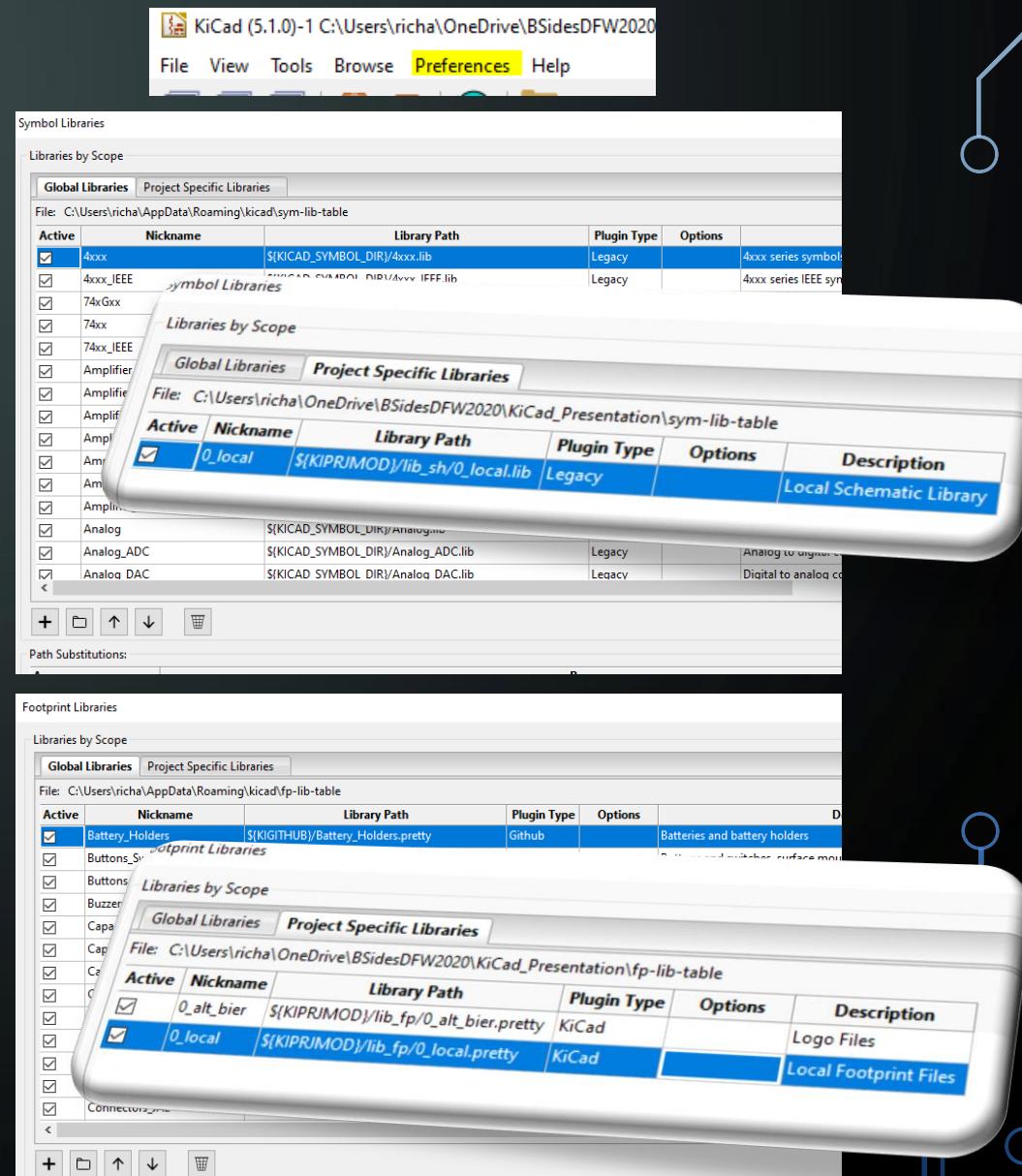
Here is a list of those files and a recommended directory structure to use for organization.

- < project name >.pro – the main project file.
Holds parameters that apply to the entire project (e.g. schematic, PCB layout, etc.).
- < project name >.sch – a schematic file.
- < project name >.kicad_pcb - a PCB New layout file.
- < project name >.net – netlist in "Pcbnew" format
- < project name >.bak – backup of schematic file.
- < project name >.kicad_pcb-bak – backup of the new PCB layout file.
- < project name >-cache.lib – a local cache copy of all the symbols used in the schematic
- < project name >-kicad_wks – page layout worksheet template file
- fp-lib-table – Footprint library list
- sym-lib-table – Schematic library list
- ./3d_models/ – Optional directory contains local 3d model files
- ./gerber/ – Optional directory contains gerber formatted output files for manufacturing
- ./lib_fp/ – Optional directory contains local footprint module directories
- ./lib_sh/ – Optional directory contains local schematic library files
- ./snapshots/ – Optional directory contains prints of schematic / cad files and images

Setting Project Preferences

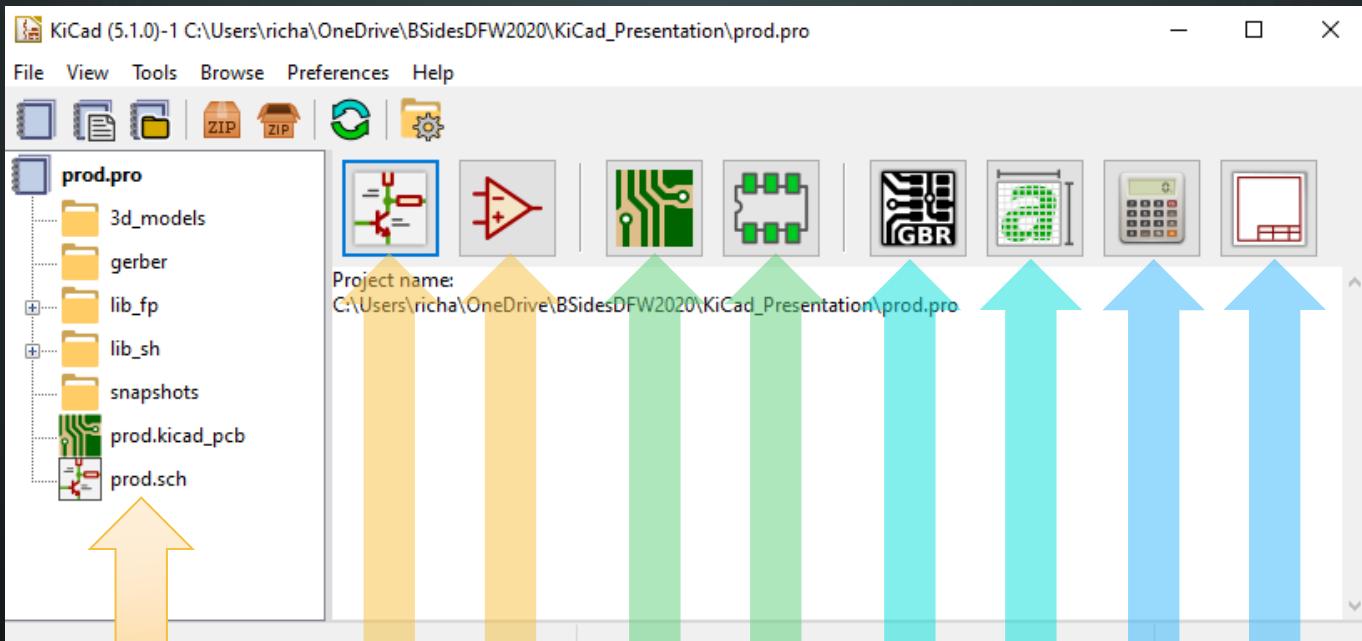
You will want to review some default project setting and adjust them as needed.

- Configure Paths allows you to set which directories will be searched for files
- Manage Symbol Libraries allows you to set which global libraries will be available to your project and the location of local project specific libraries (e.g. ./lib_sh/local.lib)
- Manage Footprint Libraries allows you to set which global libraries will be available to your project and the location of local project specific libraries (e.g. ./lib_fp/local.pretty)



KiCad Program Layout

The program layout is intuitive and easy to navigate.



Project
File
Explorer

Schematic
Layout
Editor
Eeschema

Symbol
Editor

PCB
Layout
Editor

Pcbnew

Footprint
Editor

Gerber
File
Viewer

GerbView

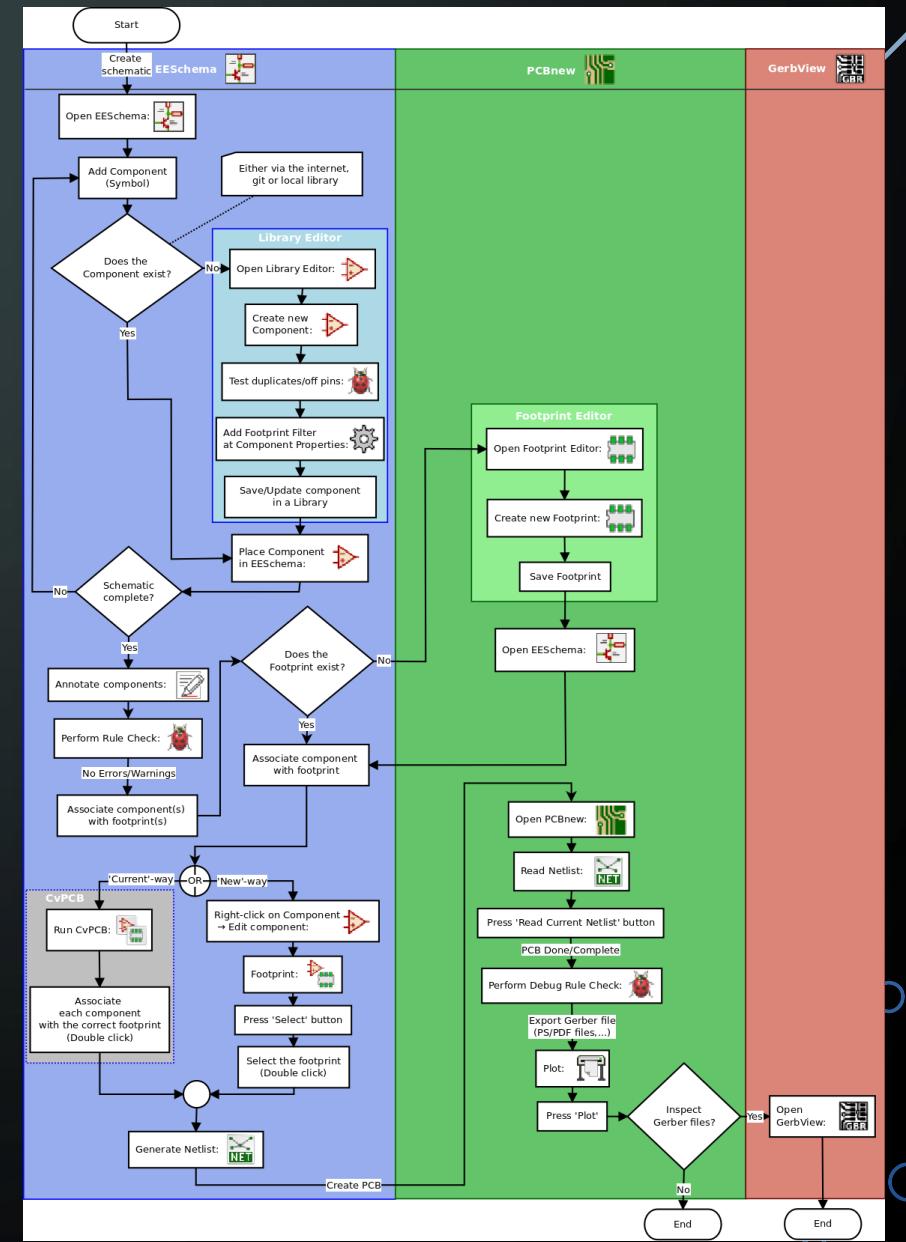
Bitmap
Converter

PCB &
Component
Calculator

Page Layout
Editor

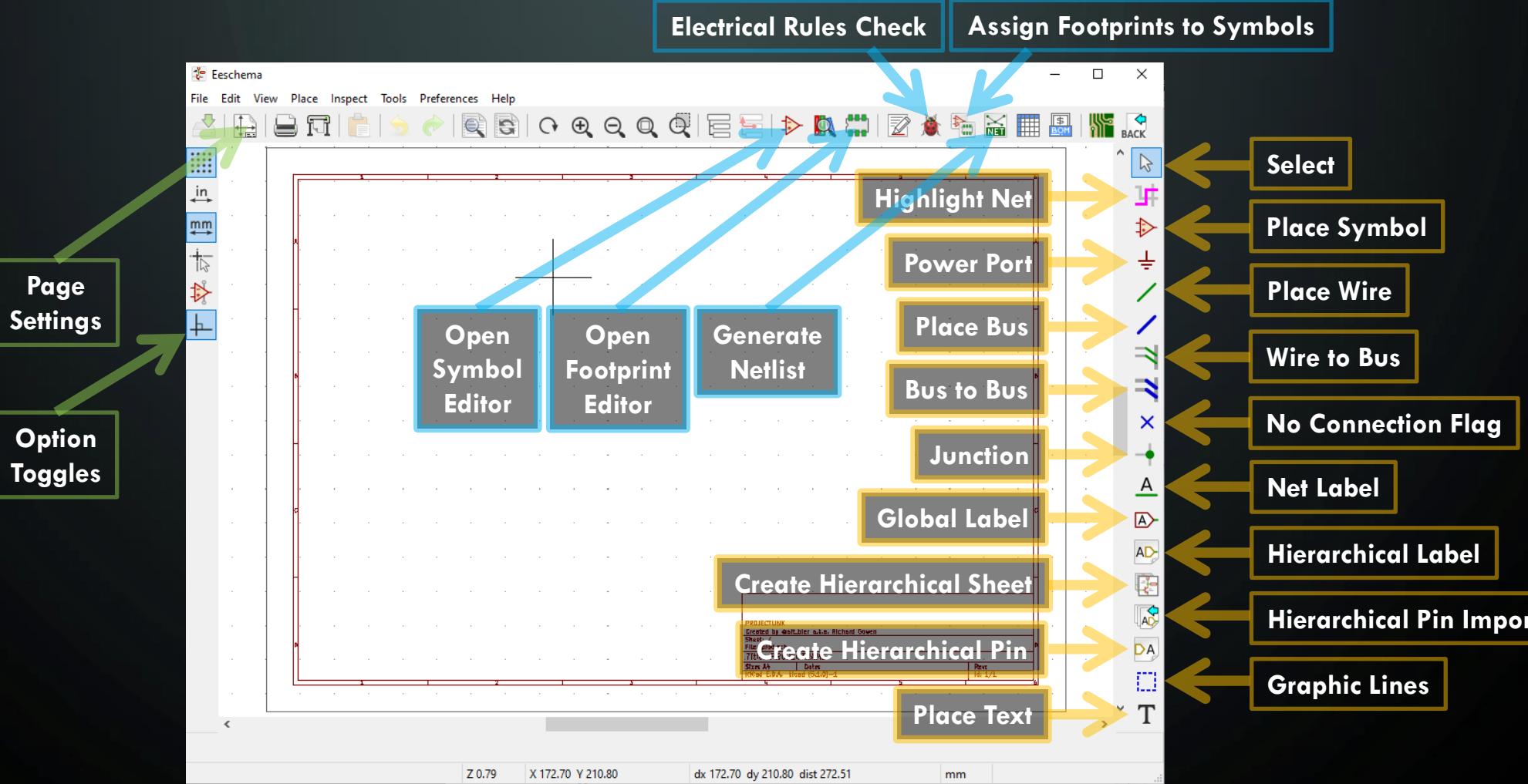
KiCad Design Workflow

- Open Eeschema to create a schematic
 - Use the symbol editor as needed to create or modify symbols
 - Use the calculator as needed to verify component values.
 - Associate component symbols with footprints
 - Use the footprint editor if needed to create or modify footprints
 - Generate a Netlist file from Eeschema
- Open Pcbnew to layout the PCB
 - Import the Netlist file
 - Size the board, move components around, run traces, add vias, etc.
 - Use the calculator as needed to verify track widths and spacing.
 - Associate component footprints with 3D models if needed
 - Use the 3D viewer to visually inspect your work.
 - Plot gerber files from Pcbnew for manufacturing
- Open GerbView to confirm the integrity of the Gerber files
- Send Gerber files to manufacturer



Eeschema Schematic Layout Editor

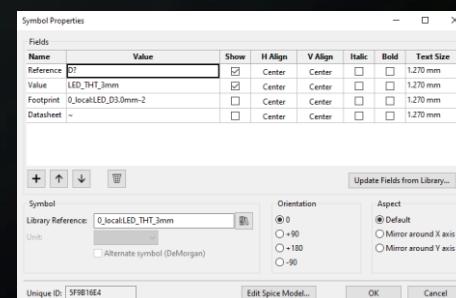
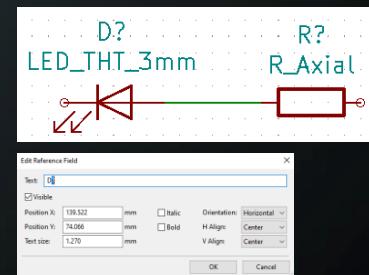
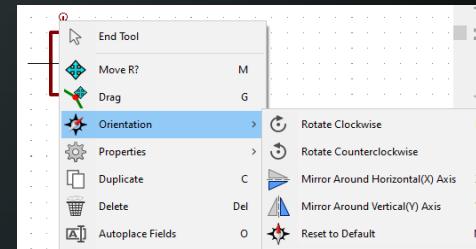
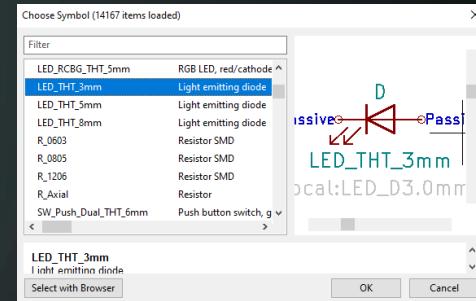
Eeschema Button Quick Reference



Eeschema Schematic Layout Editor

Eeschema Basics

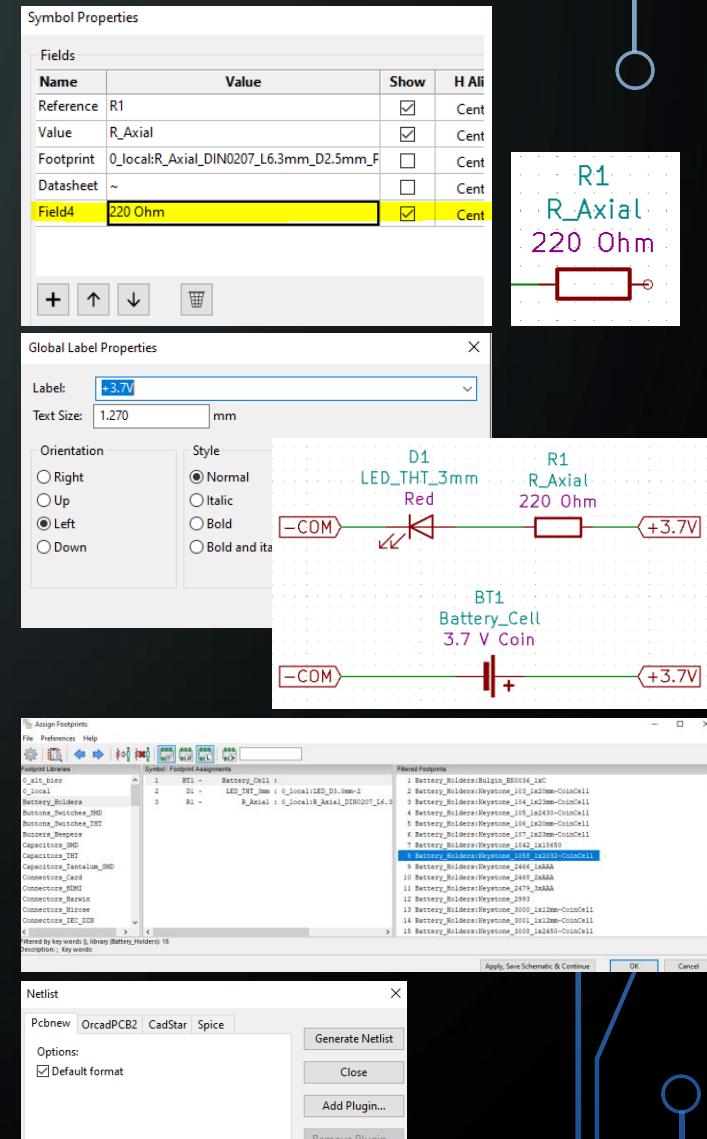
- To place a new symbol on your schematic click the Place Symbol tool and click on the schematic page. You will be prompted to choose a symbol from the available libraries.
- To move or rotate a placed symbol right-click it and choose the move or orientation menus
- To connect symbols choose the wire or bus tool and select the symbol pins to connect.
- To edit a symbol reference number double-click on it
- To edit all the properties of a symbol right click and choose the Edit Properties menu



Eeschema Schematic Layout Editor

Eeschema Basics

- To add additional information to a symbol under edit properties click the + add field and enter your text and choose if it should be shown.
- To place a label click the global label tool and click on the schematic. You will be prompted enter the label and other setting like orientation.
- Global labels can be used to connect wire nets without crossing wires on the schematic.
- Make sure that all your symbols have associated footprints by clicking the Assign Footprints button and selecting footprints for the symbols that need them.
- When complete click the Generate Netlist button to create the file that will be imported into Pcbnew

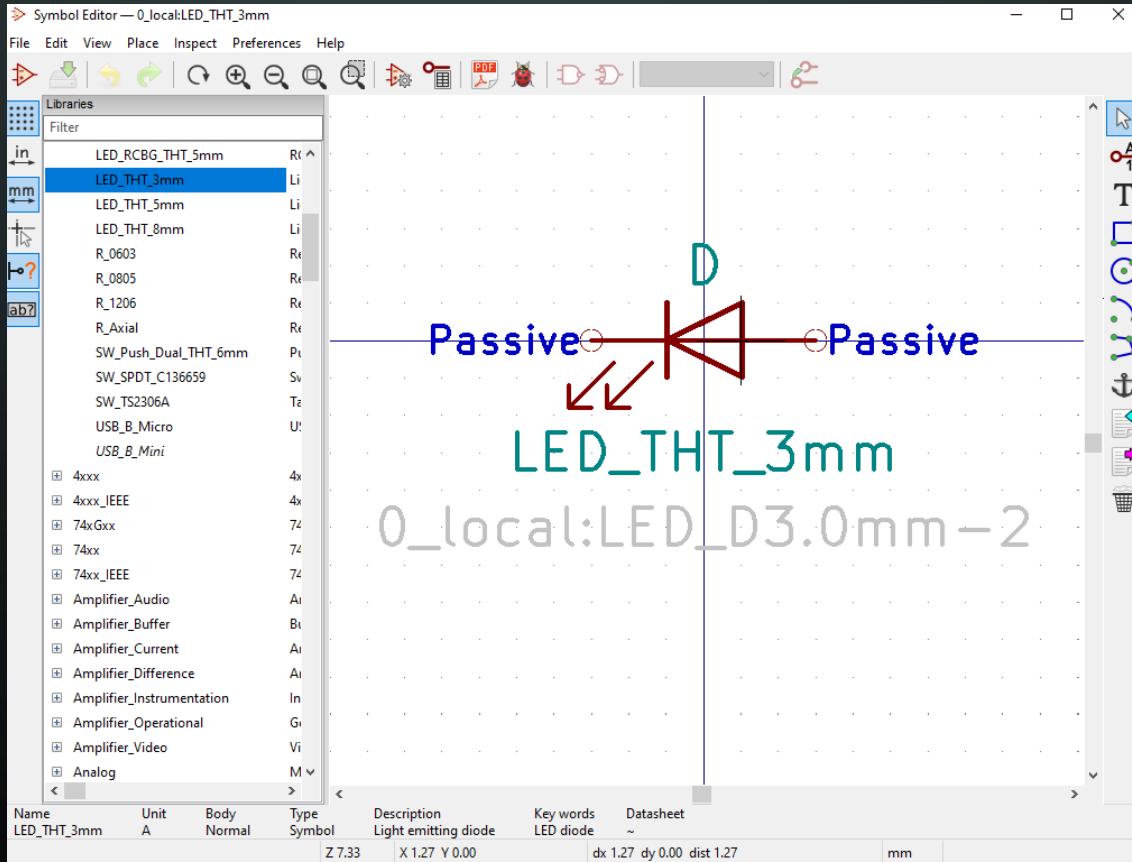


Symbol Editor

The Symbol Editor works much like the Eeschema Schematic Layout Editor. The major difference being that you are working with one symbol in a library at a time.

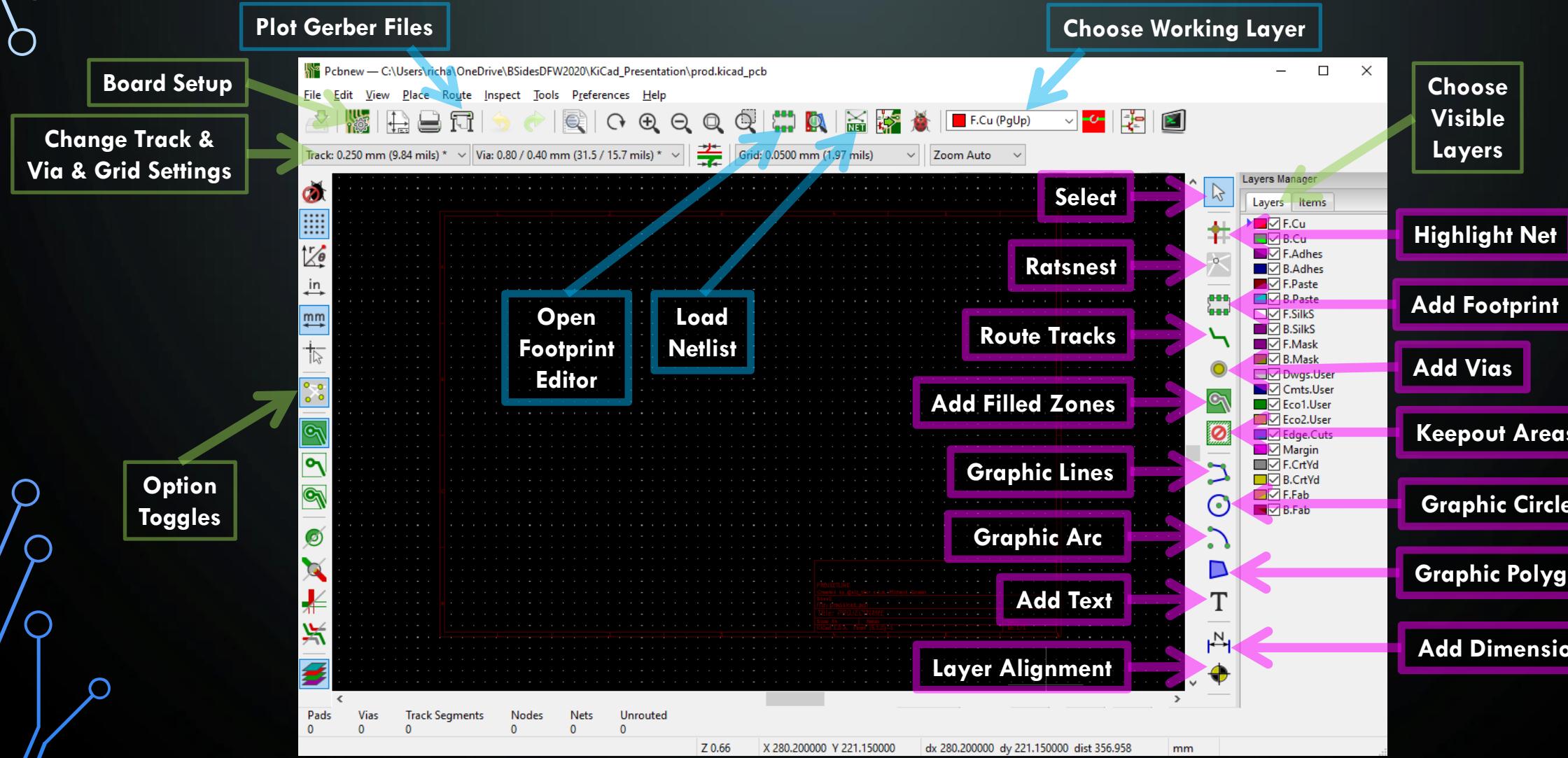
You can copy symbols to have variations of text or footprint assignments.

These symbols are stored in the libraries that are available for use by Eeschema.



Pcbnew PCB Layout Editor

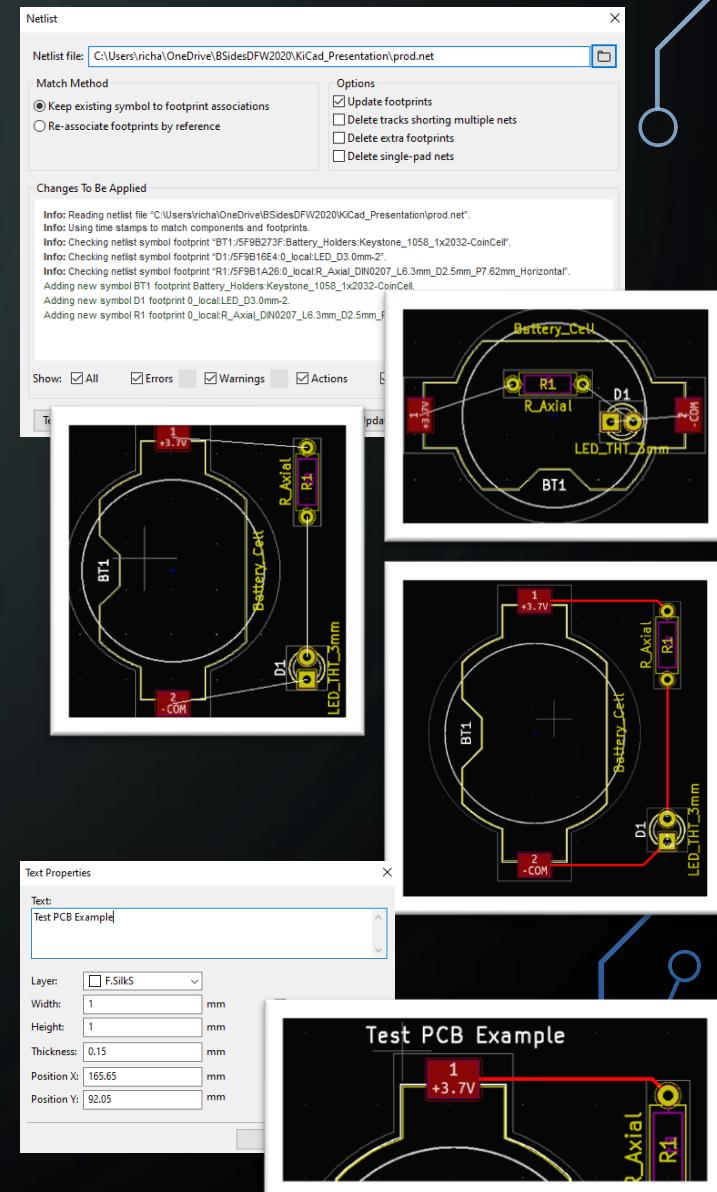
Pcbnew Button Quick Reference



Pcbnew PCB Layout Editor

Pcbnew Basics

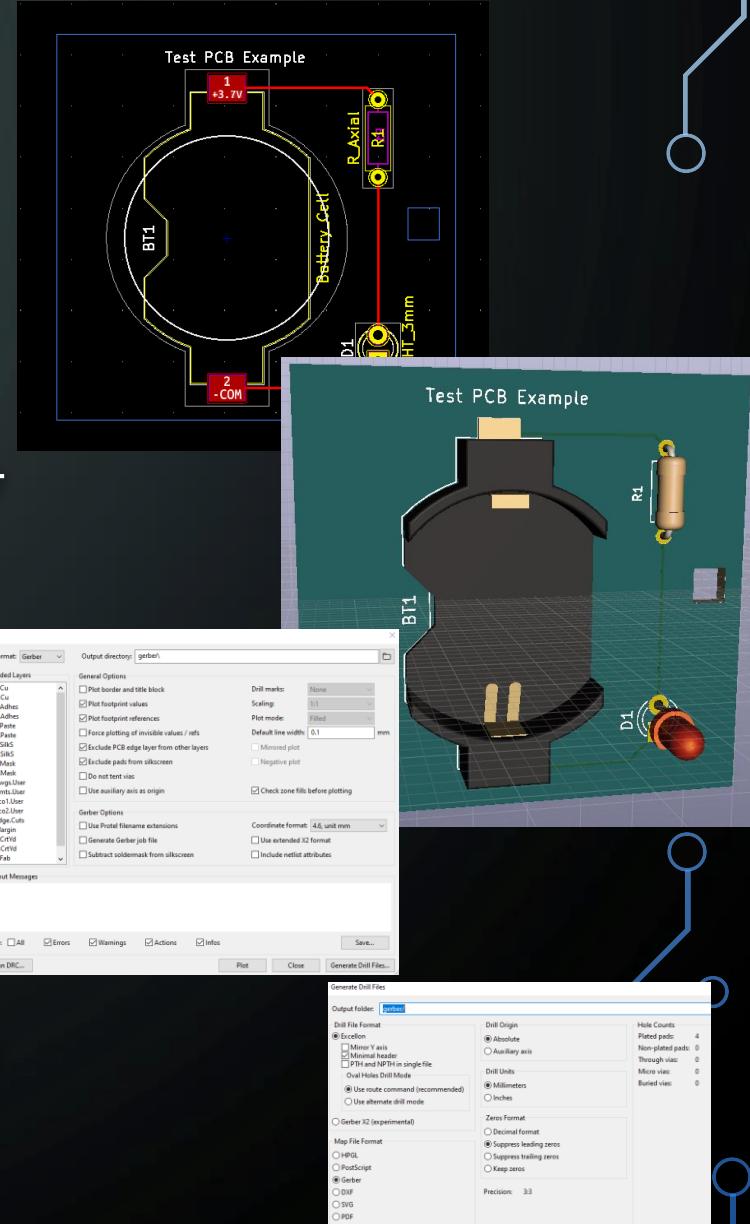
- When starting a new layout or to update a layout from schematic changes you will click the Load Netlist button to load all the schematic components. This will prompt you for the file and how you want to handle existing footprints.
- This will load or update the footprints in your layout showing lines connecting them where traces will need to be.
- Using the select tool and the properties settings move and orient your footprints where you want them on the board
- Select a copper layer as the active layer and click on route tracks to run traces between the footprint pads
- To add silkscreen text to the board select front silk as the active layer and click the add text button and click the board. This will prompt you for the text and settings. You can move this around on the PCB as needed after creation.



Pcbnew PCB Layout Editor

Pcbnew Basics

- To size your PCB you need to give it an outline in the edge cuts layer. Select Edge Cuts as the active layer and click the Add Graphic Lines tool. Then draw your outline surrounding your component footprints.
- The Edge Cuts layer can also define cut outs in the PCB.
- To visually inspect your work you can use the 3D viewer by selecting it under the view menu.
- This provides an interactive 3D render of your PCB.
- When you are done click on Plot to generate Gerber files for manufacturing. The options you choose here will be dictated by the manufacturer you are planning to use.
- In addition to the Gerber formatted design file it is usually necessary to also produce additional drill files. This is done within the plot menu by choosing the Generate Drill Files button.

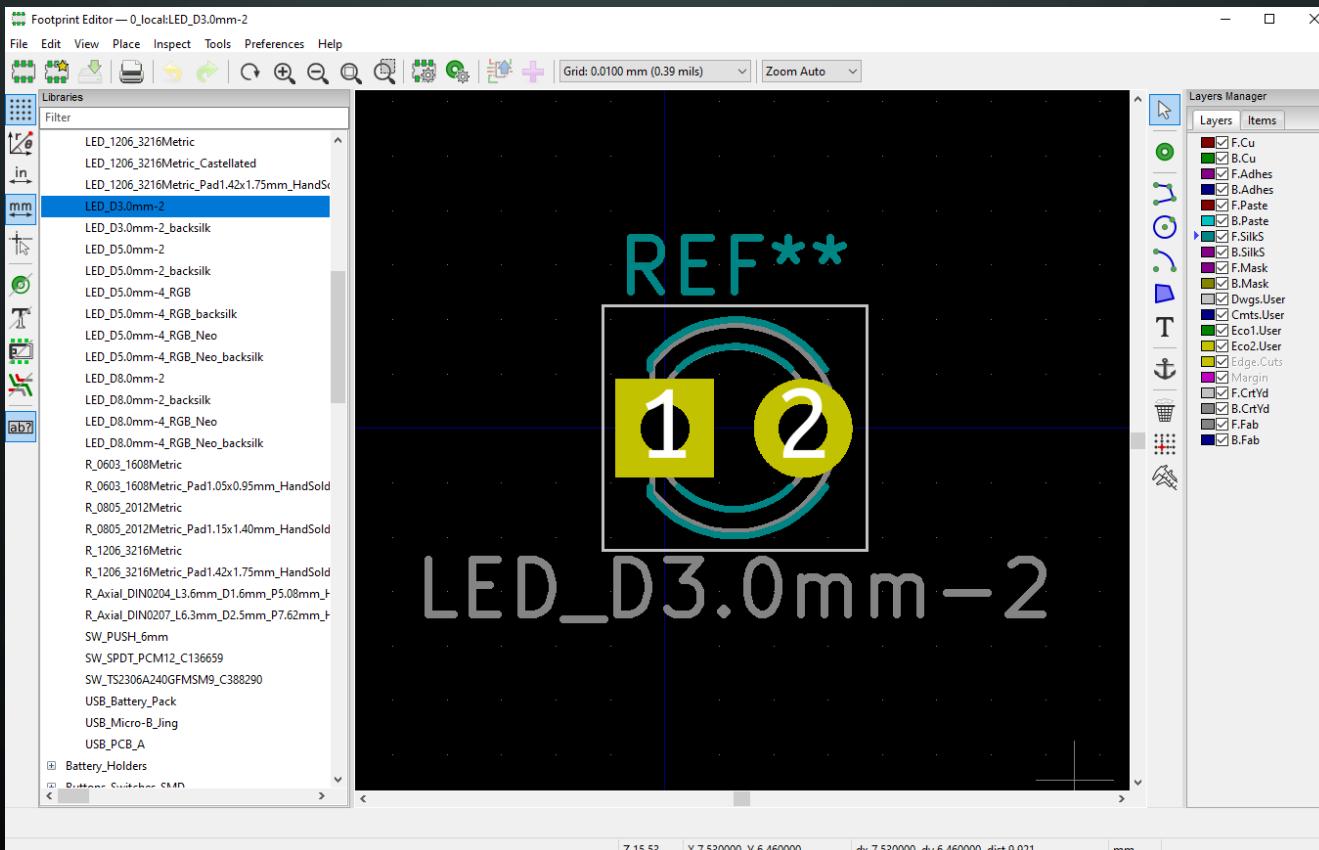


Footprint Editor

The Footprint Editor works much like the Pcbnew PCB Layout Editor. The major difference being that you are working with one footprint in a library at a time.

You can copy footprints to have variations of text or 3D model assignments.

These footprints are stored in the libraries that are available for use by Pcbnew.

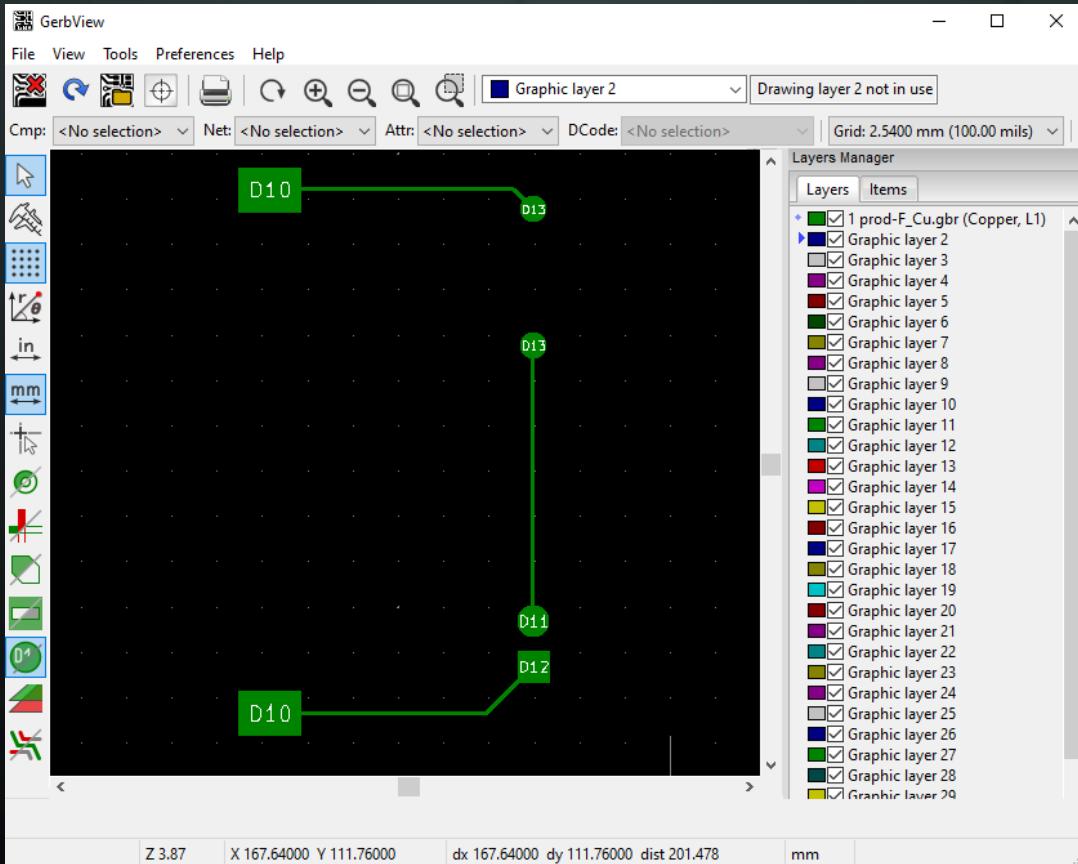


Gerber File Viewer

The Gerber File Viewer lets you load Gerber formatted design files for viewing.

These can be files you have created or from a project you downloaded.

There is a separate Gerber file for each layer of the design. You can open multiple files at once to overlay the entire PCB design.

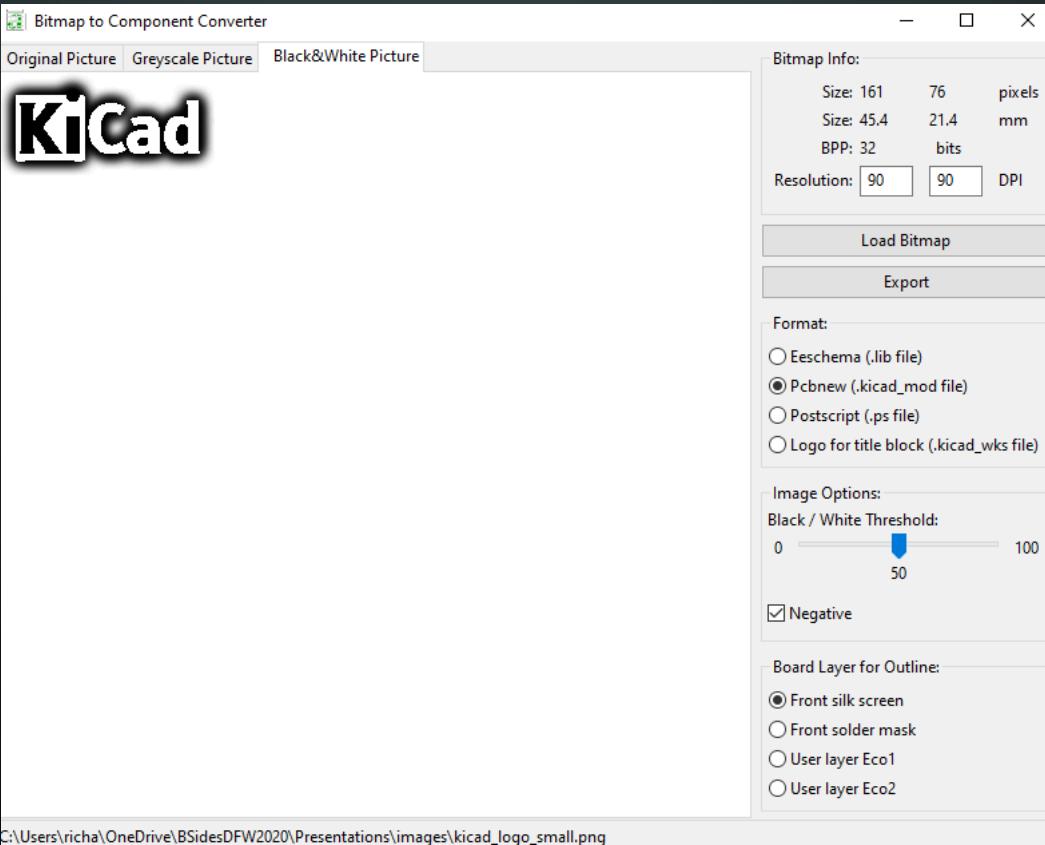


Bitmap to Component Converter

This tool takes a bitmap image to generate a footprint file.

This can be a handy way to import a logo to your silkscreen layer.

It's also good for complex fill areas or edge cuts created in an art program.



PCB Calculator

This tool bundles several useful calculation and reference items.

Regulators

Parameters:

- Current: **1.0** A
- Temperature rise: **10.0** deg C
- Conductor length: **20** mm
- Resistivity: **1.72e-8** Ohm-meter

If you specify the maximum current, then the trace widths will be calculated to suit. If you specify one of the trace widths, the maximum current it can handle will be calculated. The width for the other trace to also handle this current will then be calculated. The controlling value is shown in bold.

The calculations are valid for currents up to 35A (external) or 17.5A (internal), temperature rises up to 100 deg C, and widths of up to 400mil (10mm). The formula, from IPC 2221, is $I = K * dT^{0.44} * (W^*H)^{0.725}$

where:
I = maximum current in amps
dT = temperature rise above ambient in deg C
W,H = width and thickness in mils
K = 0.024 for internal traces or 0.048 for external traces

External layer traces:

Trace width:	0.300387 mm
Trace thickness:	0.035 mm
Cross-section area:	0.0105135 mm x mm
Resistance:	0.0327197 Ω
Voltage drop:	0.0327197 Volt
Power loss:	0.0327197 Watt

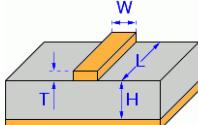
Internal layer traces:

Trace width:	0.781437 mm
Trace thickness:	0.035 mm
Cross-section area:	0.0273503 mm x mm
Resistance:	0.0125776 Ω
Voltage drop:	0.0125776 Volt
Power loss:	0.0125776 Watt

TransLine

Transmission Line Type:

- Microstrip Line
- Coplanar wave guide
- Coplanar wave guide with ground plane
- Rectangular Waveguide
- Coaxial Line
- Coupled Microstrip Line
- Stripline
- Twisted Pair



Substrate Parameters:

Er:	4.6
TanD:	0.02
Rho:	1.72e-08
H:	0.2 mm
H_t:	1e+20 mm
T:	0.035 mm
Rough:	0
mu Rel S:	1
mu Rel C:	1

Physical Parameters:

W:	0.2 mm
L:	50 mm

Analyze | **Synthesize**

Electrical Parameters:

Z0:	50 Ω
Ang. I:	0 Radian

Results:

- ErEff:
- Conductor Losses:
- Dielectric Losses:
- Skin Depth:

RF Attenuators

Attenuators:

- Pi
- Tee
- Bridged Tee
- Resistive Splitter

Parameters:

Attenuation	6 dB
Zin	50 Ω
Zout	50 Ω

Formula

Z_{in} desired input impedance in Ω

Z_{out} desired output impedance in Ω

a attenuation in dB

$L = 10^{a/10}$ (the loss)

$A = (L + 1)/(L - 1)$

Pi attenuator

$R2 = (L - 1)/2 * \sqrt{((Z_{in} * Z_{out})/L)}$

Values:

R1	Ω
R2	Ω
R3	Ω

Messages:

Color Code

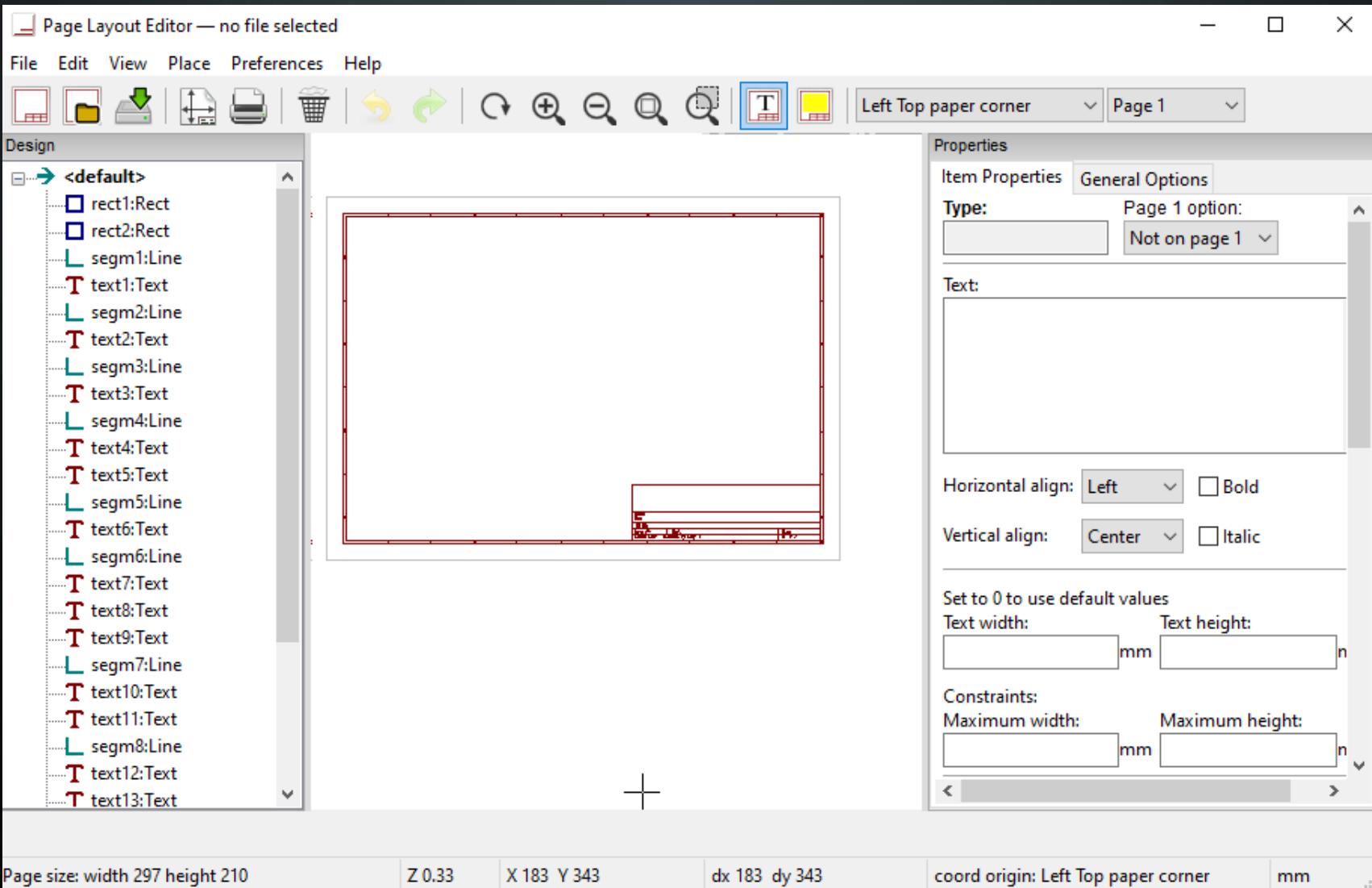
1st Band	2nd Band	3rd Band	4th Band	Multiplier	Tolerance
Black	0	0	0	x 1	± 1%
Brown	1	1	1	x 10	± 2%
Red	2	2	2	x 100	
Orange	3	3	3	x 1k	
Yellow	4	4	4	x 10k	
Green	5	5	5	x 100k	± 0.5%
Blue	6	6	6	x 1M	± 0.25%
Violet	7	7	7	x 10M	± 0.10%
Grey	8	8	8	x 100M	± 0.05%
White	9	9	9	x 1G	
Gold				x 0.1	± 5%
Silver				x 0.01	± 10%

Tolerance

- 10% / 5%
- <= 2%

Page Layout Editor

This tool allows you to change the default look of the printed design pages.



PCB Manufacturing

Once you have completed your design you are ready to send it to be manufactured.

There are several factors to consider when choosing a PCB manufacturer. Cost is probably the top factor. But time and location and quality are all also important. The cheapest PCB manufacturer might take months to deliver or might have poor quality.

Look at customer reviews and do some small prototype runs to confirm they meet your needs. There are many PCB manufacturers to choose from.

Here are a few:

- <https://pcbshopper.com/>  Travelocity like site for PCB Mfrs.
- <https://jlcpcb.com/>  Very Inexpensive but with some limitations.
- <https://pcbway.com/>  Inexpensive and fast shipping.
- <https://oshpark.com/>  Beautiful, High Quality, made in USA

THANK YOU

I hope you enjoyed this presentation and learned something from it.

-- @alt_bier

This Slide Deck – <https://altbier.us/pcb/>

Code – https://github.com/gowenrw/BSidesDFW_2020_HHV/