

Introduction to For Circuit Board Design

Dave Maciorowski, WA1JHK
wa1jhk@macski.us

jhk-labs.com
17 October 2025



References

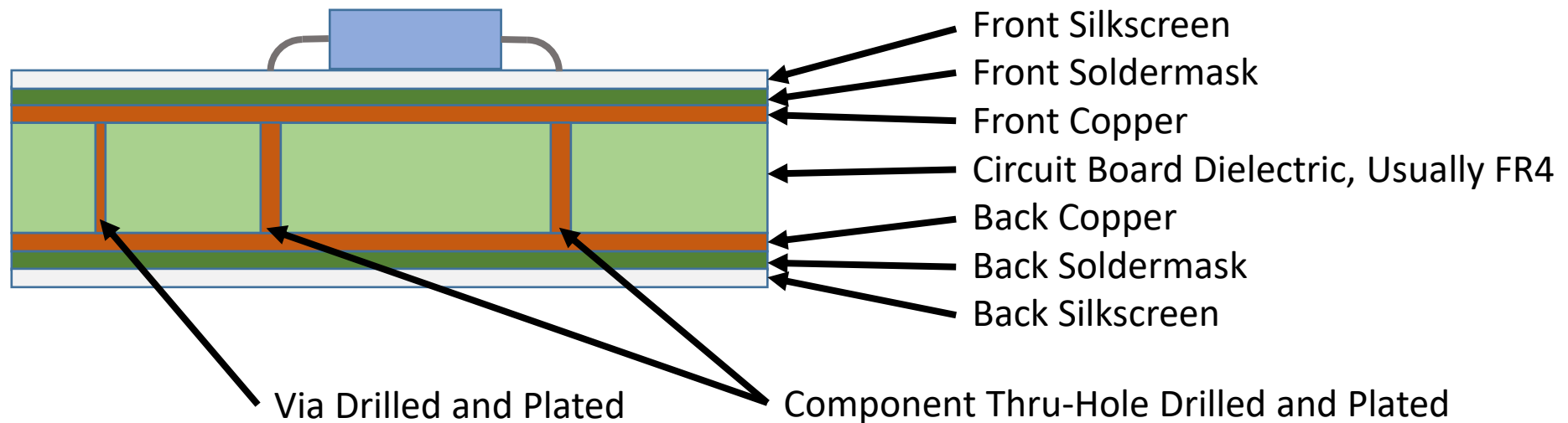
- KiCad Website
 - <https://www.kicad.org/>
- KiCad Tutorials – Watch for your KiCad Version
 - Lots of tutorials on <https://www.youtube.com/>
- JLCPCB Reference
 - <https://jlcpcb.com/help/article/how-to-generate-gerber-and-drill-files-in-kicad-8>
- Misc
 - https://en.wikipedia.org/wiki/Gerber_format
- Digital Caliper
 - <https://www.amazon.com/dp/B017KUC6XQ>

Topics

- Anatomy of a PCB
- Work Flow – How to use the Tools
- KiCad Main – Finding Your Way Around
- KiCad -- Schematic Layout Editor
- KiCad -- PCB Layout Editor
- KiCad – Generating Gerbers, Ordering Your First Board at JLCPCB
- References

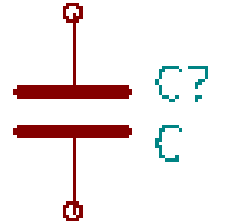
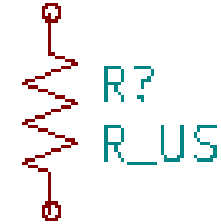
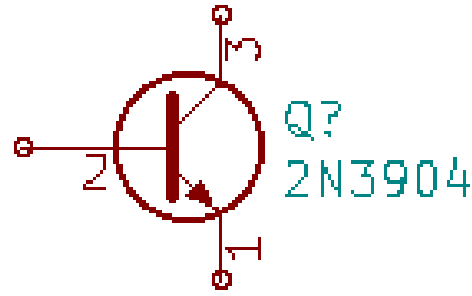
Anatomy of a PCB

- Many Steps to Manufacturing a PCB
- “Gerber” Files Used in the Manufacturing Process
- KiCad Creates Gerber Files
- “Layers” Refers to the Number of Copper Layers, Always Even, Up to 30



Anatomy of a PCB

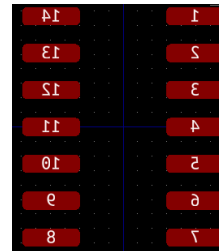
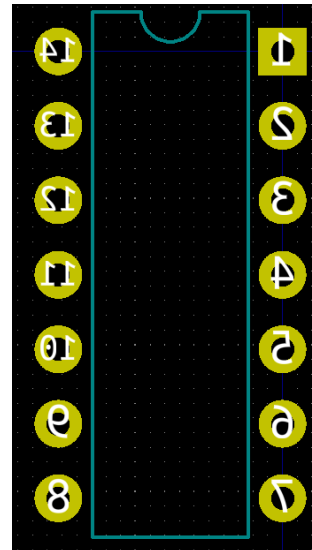
- Schematic Symbols



- Circuit Board Footprints

- ICs

- DIP-14
 - SOIC-14



- Resistors

- .3 Inch Axial



- 0603 == .06 x .03 Inch



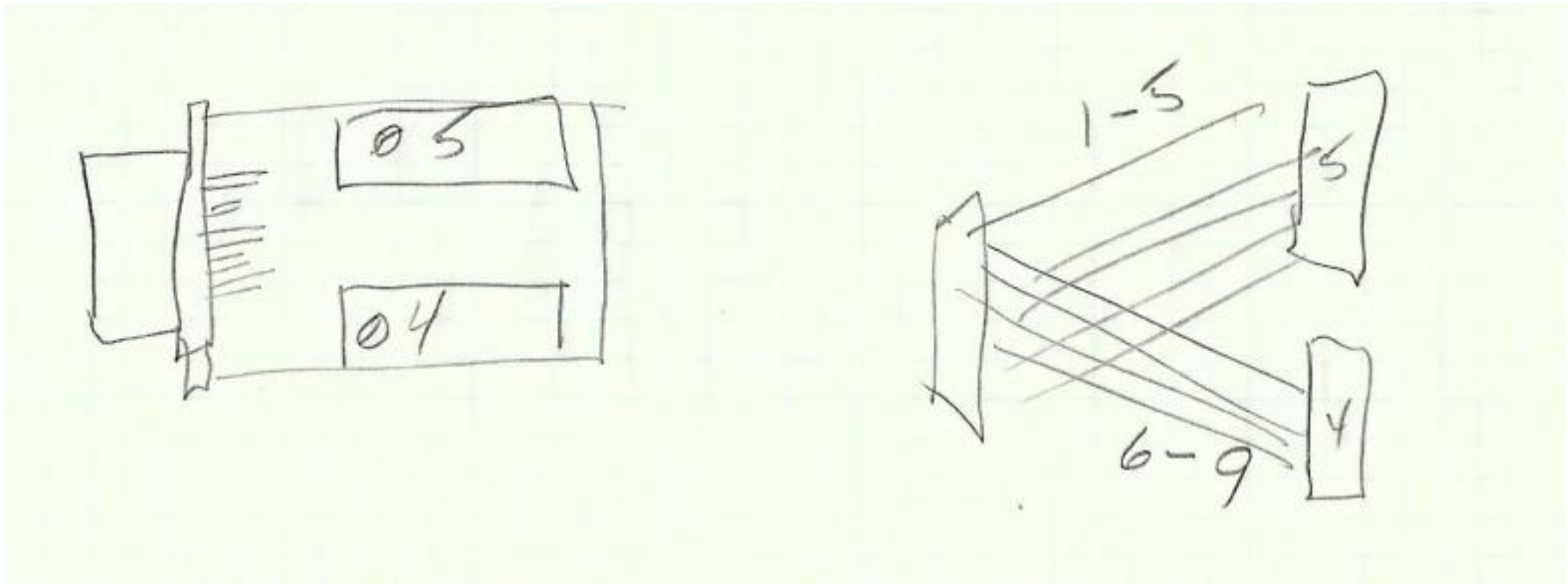
KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

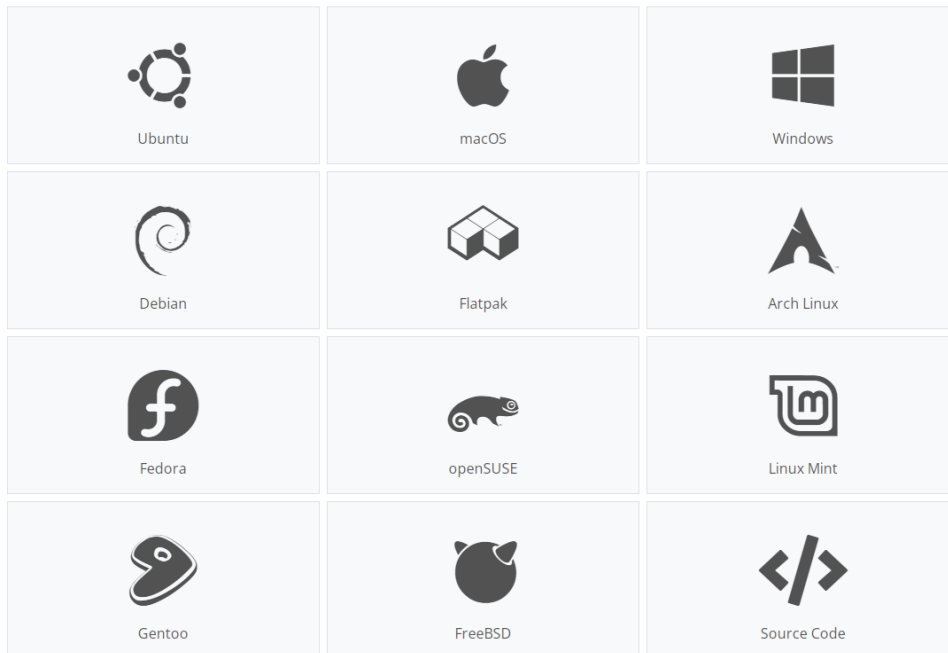
Example Project – Hand Sketch



KiCad

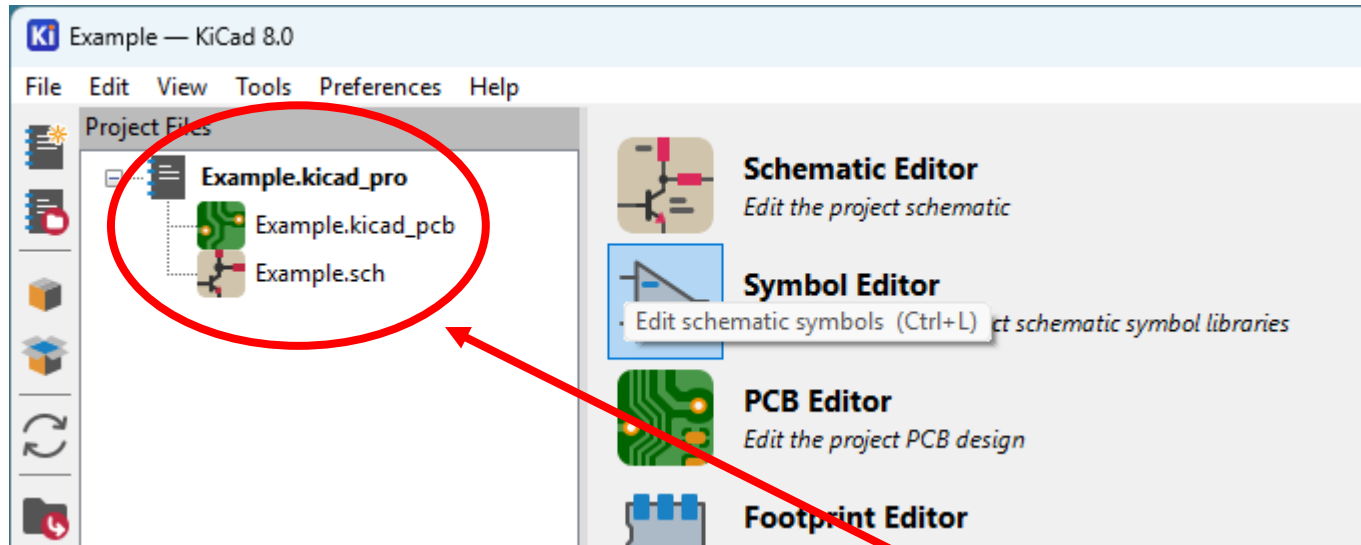
- Pronunciation – KEY-CAD
- Official Site -- <https://www.kicad.org/>
- Installing-- <https://www.kicad.org/download/>

Select your operating system or distribution



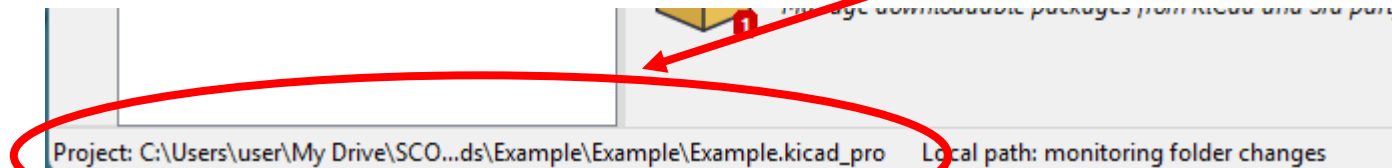
- Choose Your Platform
 - Windows
 - macOS
 - Linux in Many Flavors
 - FreeBSD
 - Source Code

KiCad – Main Screen

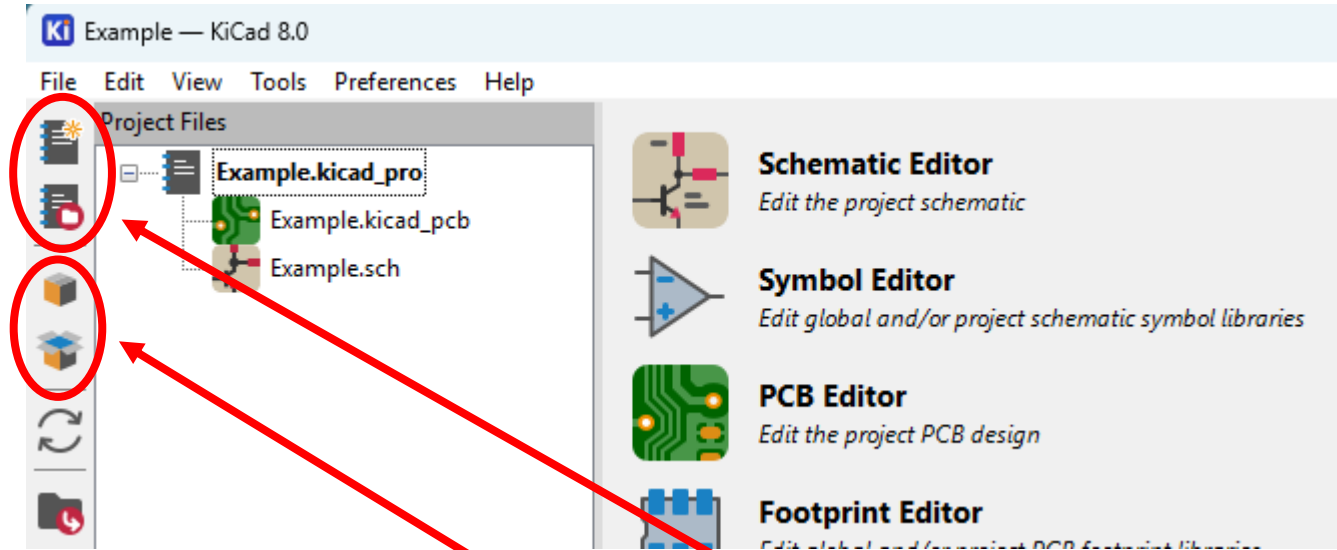


Project Files

Project Folder Location



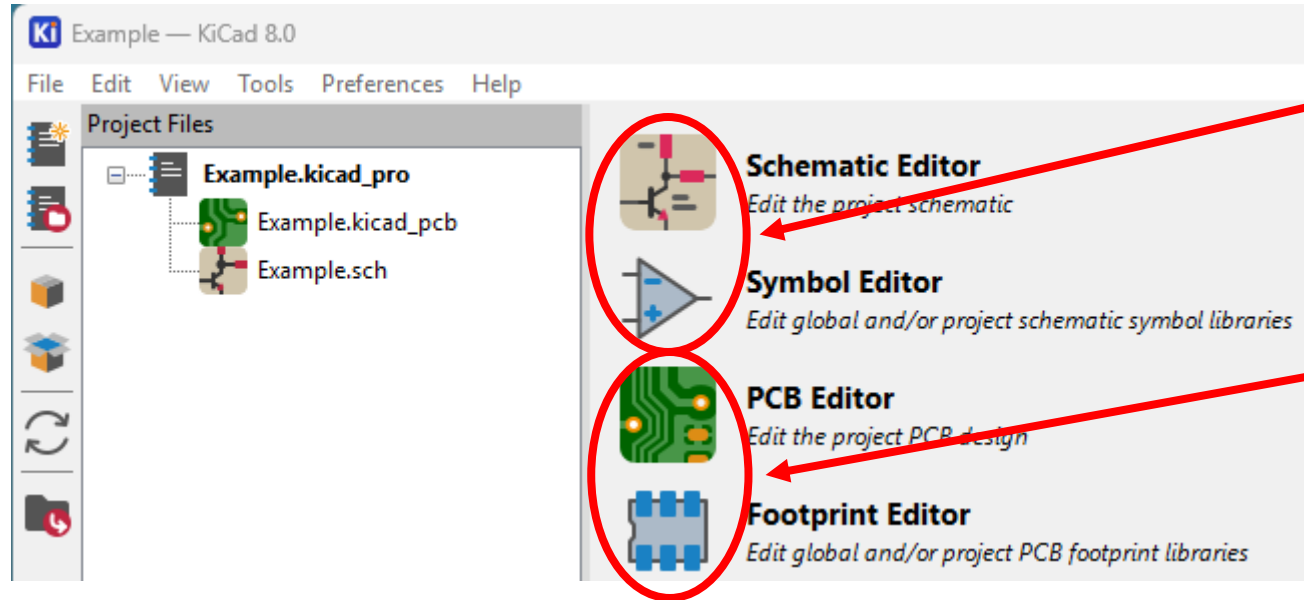
KiCad – Main Screen



Create and Open Project Files

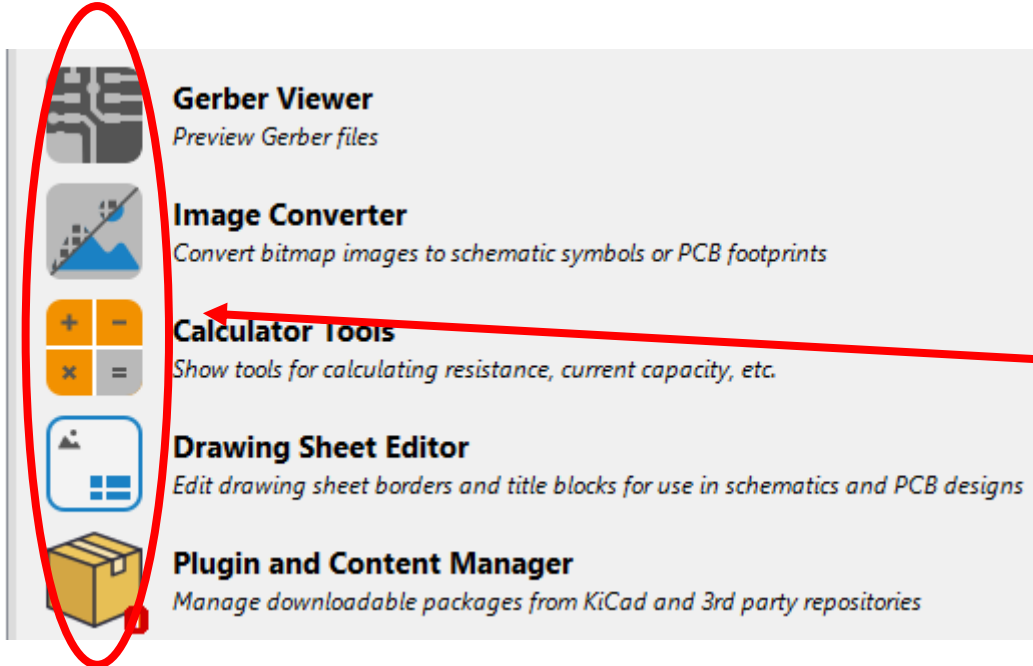
Archive and Unarchive Project

KiCad – Main Screen



- Schematic Tools
 - Schematic Layout Editor
 - Symbol Editor
- PCB Tools
 - PCB Layout Editor
 - Footprint Editor

KiCad – Main Screen

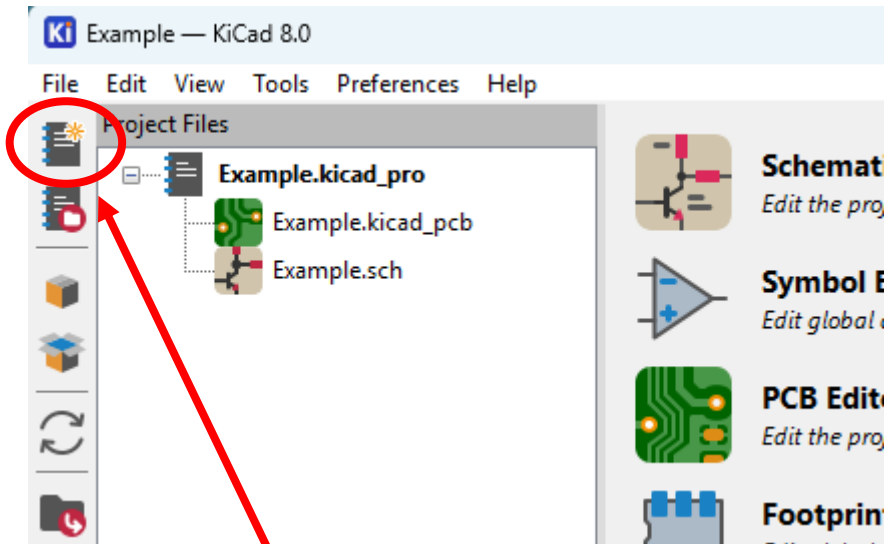


- Gerber Viewer
- Image Converter
- Calculator Tools
- Drawing Sheet Editor
- Plugin and Content Manager

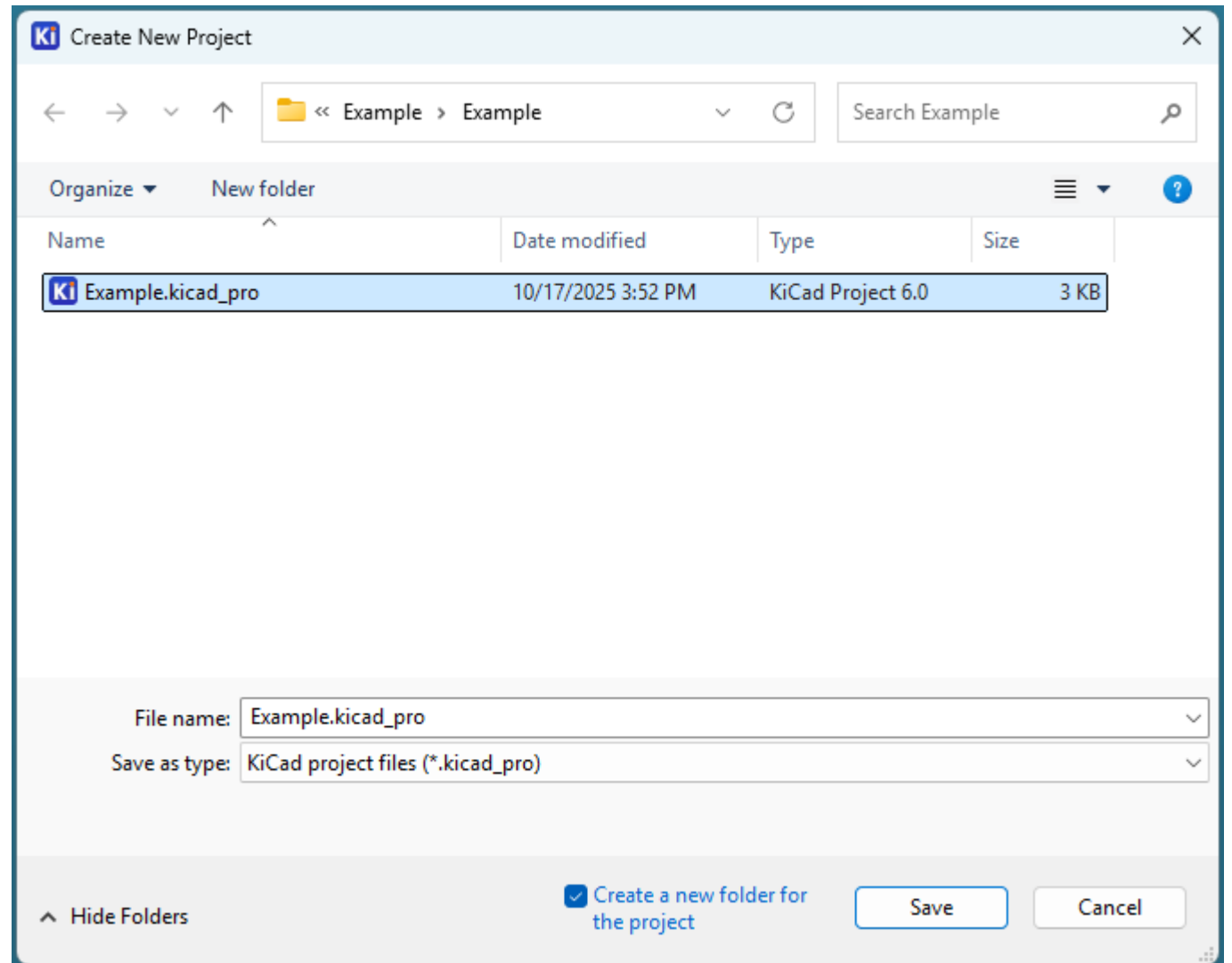
KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

KiCad – Main Screen



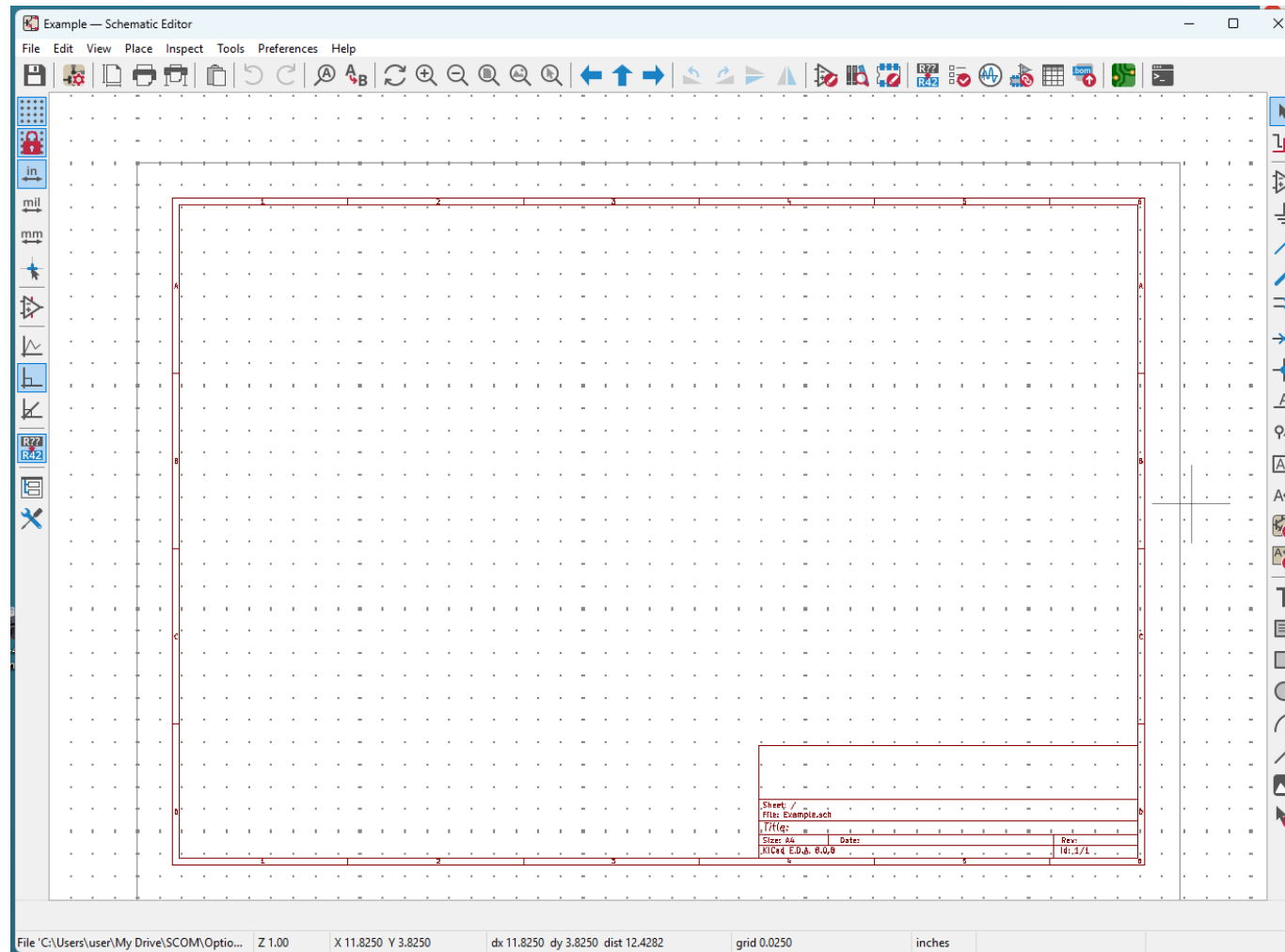
Click to Create Project



KiCad – Work Flow

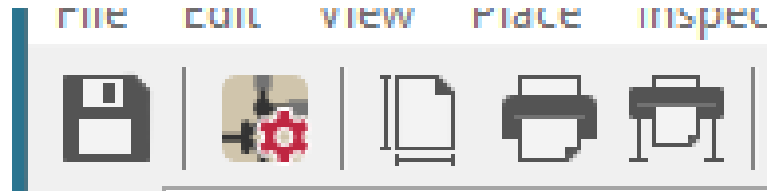
- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

Schematic Layout Editor



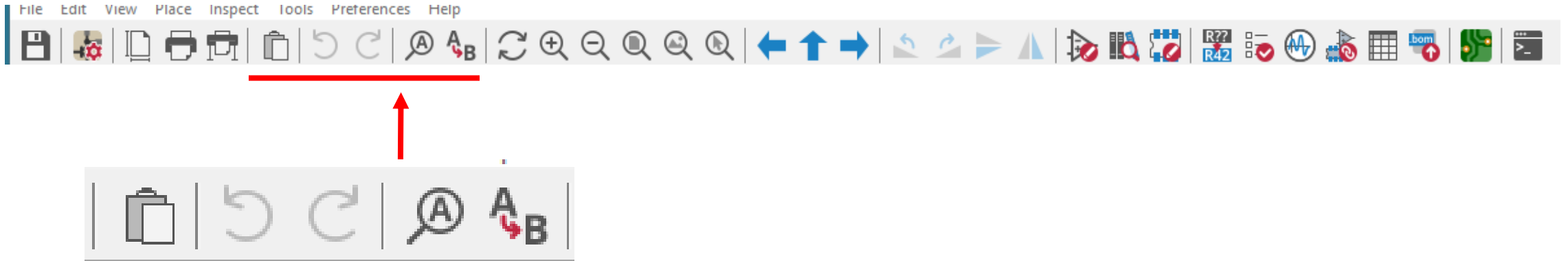
- Left – Mode Selection
- Top – Viewing and Navigation Tools
- Right – Drawing Tools

Schematic Layout Editor



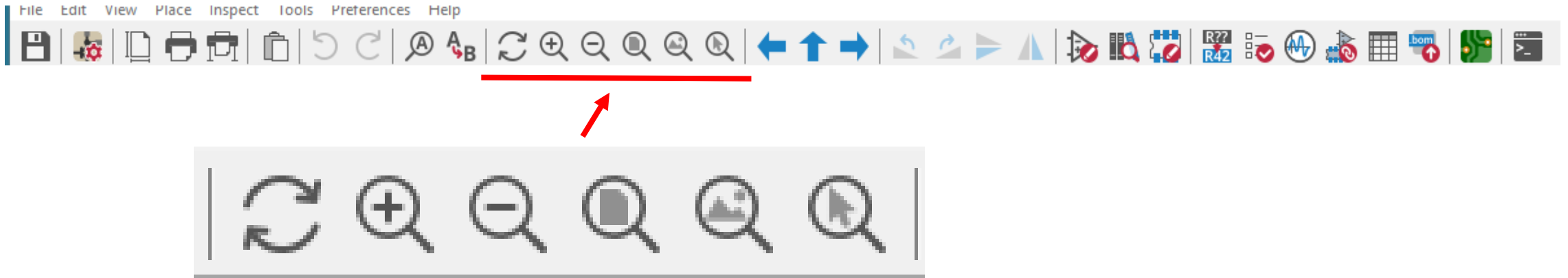
- Save All
- Schematic Setup
- Page Settings – Set Sheet Size and Title Block
- Print – Schematic to a Printer
- Plot – Schematic to a PDF

Schematic Layout Editor



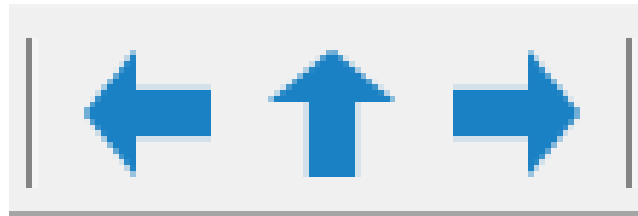
- Editing
 - Paste – CTRL+V
 - Undo – CTRL+Z
 - Redo – CTRL+Y
 - Find Symbols and Text – CTRL+F
 - Find and Replace Symbols and Text – CTRL+ALT+F

Schematic Layout Editor



- Schematic View
 - Refresh – F5
 - Zoom In – F1 or Scroll Wheel
 - Zoom Out – F2 or Scroll Wheel
 - Zoom to Fit -- HOME
 - Zoom to Objects – CTRL+HOME
 - Zoom to Selection – CTRL+F5

Schematic Layout Editor



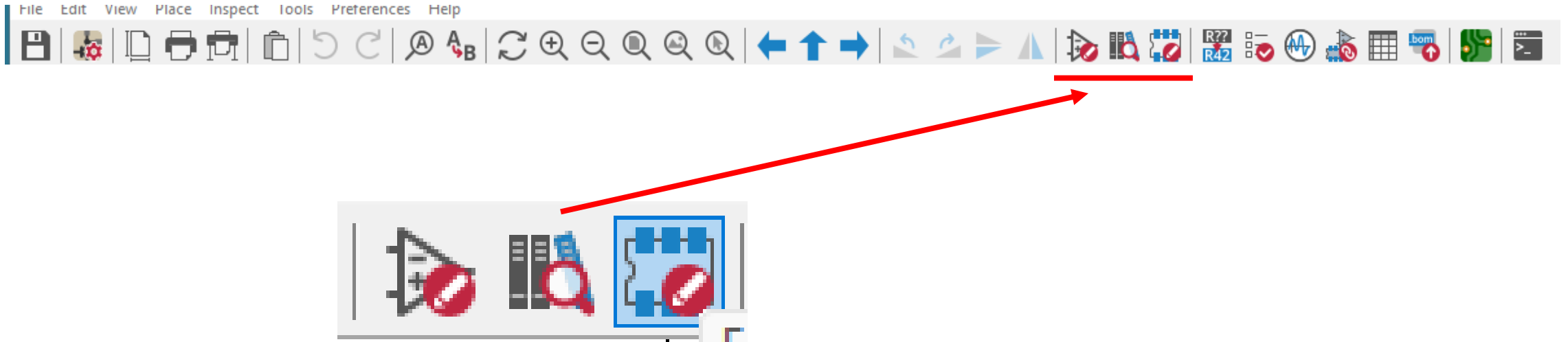
- Multiple Hierarchical Schematic Sheets
 - Navigate Backward
 - Navigate Up
 - Navigate Forward

Schematic Layout Editor



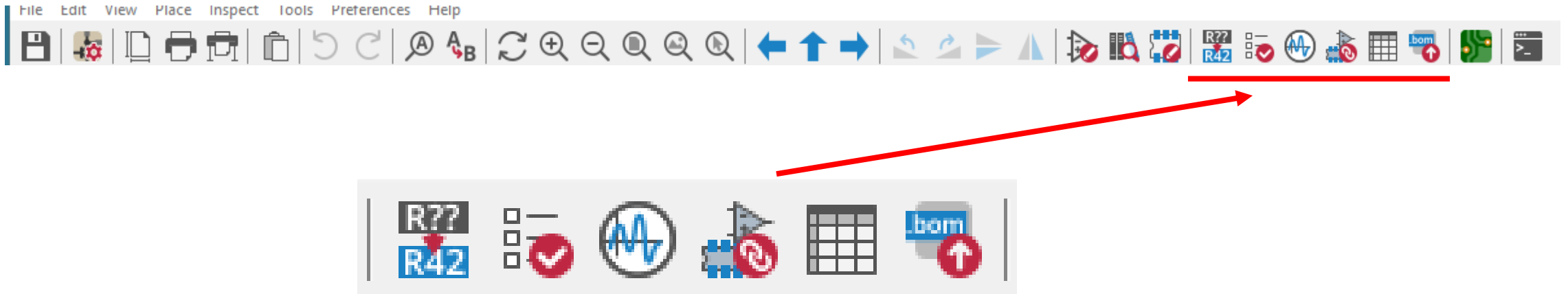
- Selected Symbol
 - Rotate Counterclockwise -- R
 - Rotate Clockwise
 - Mirror Vertically – Y
 - Mirror Horizontally -- X

Schematic Layout Editor



- Symbols and Footprints
 - Symbol Editor -- Create, Delete and Edit Symbols
 - Symbol Library Browser – *Spend Some Time Here*
 - Footprint Editor -- Create and Edit Symbols

Schematic Layout Editor



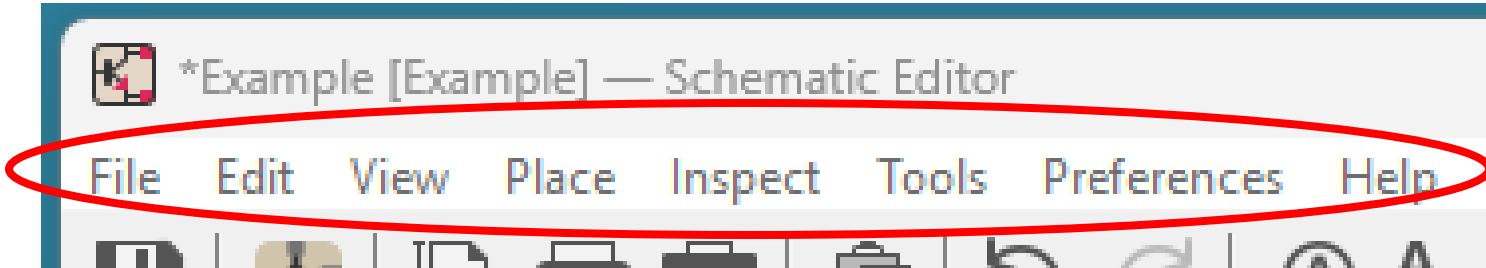
- Process Schematic
 - Annotate Schematic – Add Symbol Reference Designators
 - Electrical Rules Checker – Find Errors that Would Prevent Netlist
 - Simulator – Run SPICE or IBIS Simulations
 - Assign PCB Footprints to Schematic Symbols
 - Edit Symbol Fields – Resolve Errors from Check
 - Generate Bill of Materials – Parts List

Schematic Layout Editor



- Start the PCB Layout Editor
- Python Scripting Console

Schematic Layout Editor

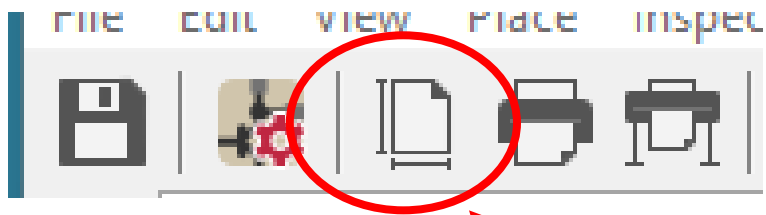


- Menu
 - File -- File Management
 - Edit – Editing Tools
 - View – Change the Schematic View
 - Place – Add Schematic Elements, e.g. Symbols, Text, Wires
 - Tools – Select a Tool
 - Preferences -- Configuration
 - Help – Documentation, List Hotkeys

KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

Schematic Layout Editor



Page Size – Set the schematic sheet size
And Title Block

Schematic Layout Editor

Set Paper Size

Set Date

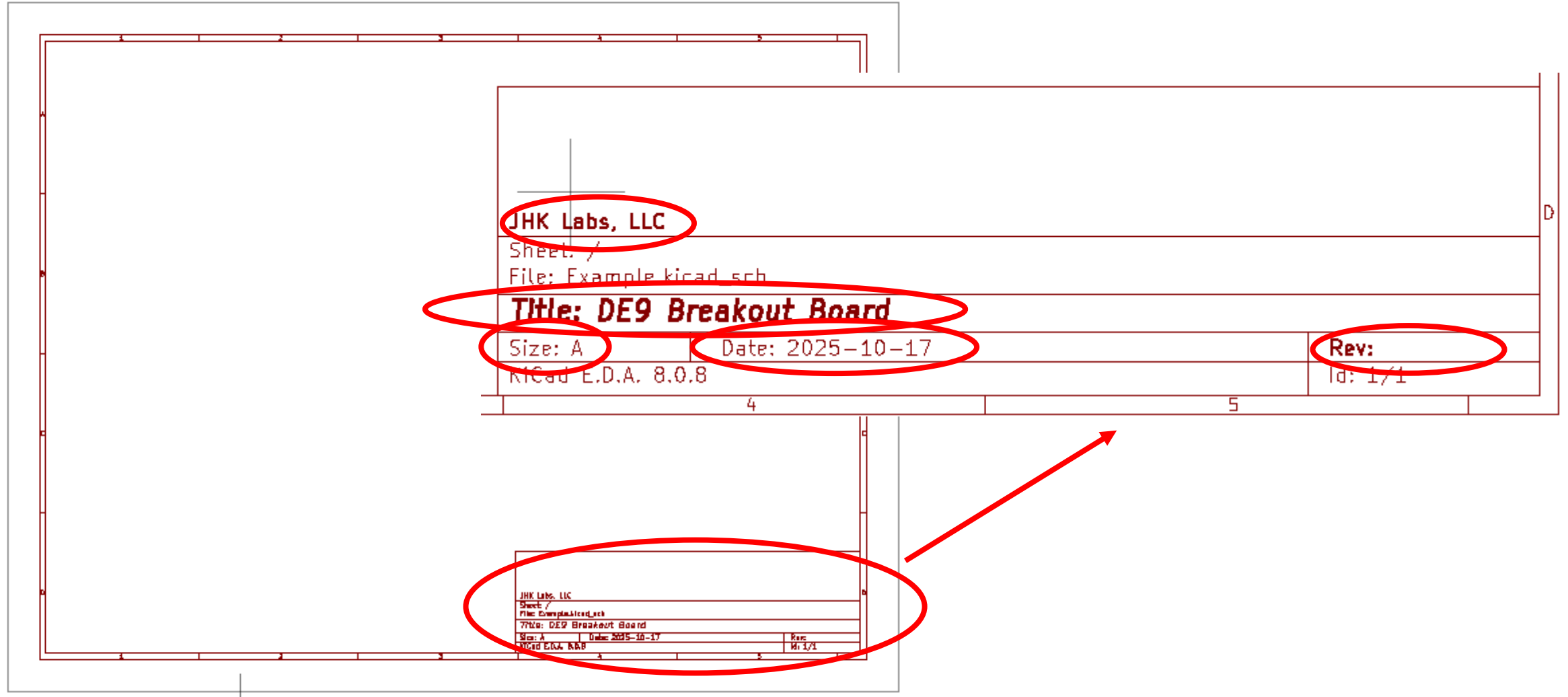
Set Title Block Info

Click OK

The screenshot shows the 'Page Settings' dialog box with the following sections and annotations:

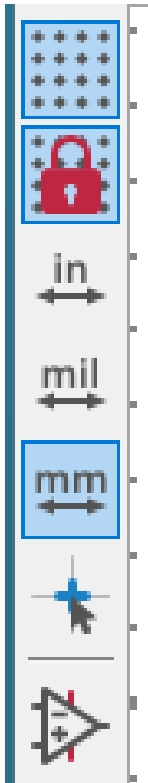
- Paper Section:**
 - Size:** A red circle highlights the 'A4 210x297mm' dropdown menu, with a red arrow pointing to it from the 'Set Paper Size' text.
 - Orientation:** A dropdown menu set to 'Landscape'.
 - Custom paper size:** Fields for Height (11 in) and Width (17 in).
 - Export to other sheets:** A checked checkbox.
 - Preview:** A small schematic drawing of a circuit board.
- Drawing Sheet Section:**
 - File:** An empty text field with a folder icon.
- Title Block Section:**
 - Number of sheets:** 1, **Sheet number:** 1.
 - Issue Date:** A date field showing '10/17/2025' with a red circle and a red arrow pointing to it from the 'Set Date' text.
 - Revision:** A text field.
 - Title:** A text field.
 - Company:** A text field.
 - Comment1:** A text field.
 - Comment2:** A text field.
 - Comment3:** A text field.
 - Comment4:** A text field.
 - Comment5:** A text field.
 - Comment6:** A text field.
 - Comment7:** A text field.
 - Comment8:** A text field.
 - Comment9:** A text field.
 - Export to other sheets:** A column of checkboxes, all of which are unchecked.
- Buttons:** At the bottom right, the 'OK' button is circled in red, with a red arrow pointing to it from the 'Click OK' text. The 'Cancel' button is also visible.

Schematic Layout Editor



Schematic Layout Editor

- Left Toolbar – Select Mode



- Turn the Grid On and Off – Blue is On
- Turn Grid Overrides On and Off – Blue is On
- Switch Units to Inches – Blue is Selected
- Switch Units to Mils – Blue is selected
- Switch Units to Millimeters -- Blue is Selected
- Choose Cursor Shape – Blue is Full Screen
- Toggle Visibility of Invisible Pins On Symbols – Blue is On

Schematic Layout Editor

- Left Toolbar – Select Mode



- Line Mode, Any Angle – Blue is Selected
- Line Mode, Horizontal or Vertical – Blue is Selected
- Line Mode, Horizontal, Vertical or 45 -- Blue is Selected
- Annotate Symbols Automatically – Blue is On
- Show Sheet Hierarchy Navigator – Blue is On
- Show Properties Manager – Blue is On

Schematic Layout Editor

- Right Toolbar



- Select Item – Click then Select Item
- Highlight Net – Click then Select Net
- Place Symbol – Click then Select and Place Symbol
- Place Power Port – Click then Select and Place Symbol
- Place Wire – Click then Place Single Connection
- Place Bus – Click then Place Bus of Connections
- Place Wire to Bus Entry – Click then Attach Wire to Bus
- Place No Connection Flag – Tell Checker it's ok not to Connect

Schematic Layout Editor

- Right Toolbar



- Place Junction – Connect Two Crossed Wires
- Place Net Label
- Place Net Class Directive
- Place Global Label
- Place Hierarchical Label
- Add Hierarchical Sheet
- Place Hierarchical Pin in Sheet

Schematic Layout Editor

- Right Toolbar

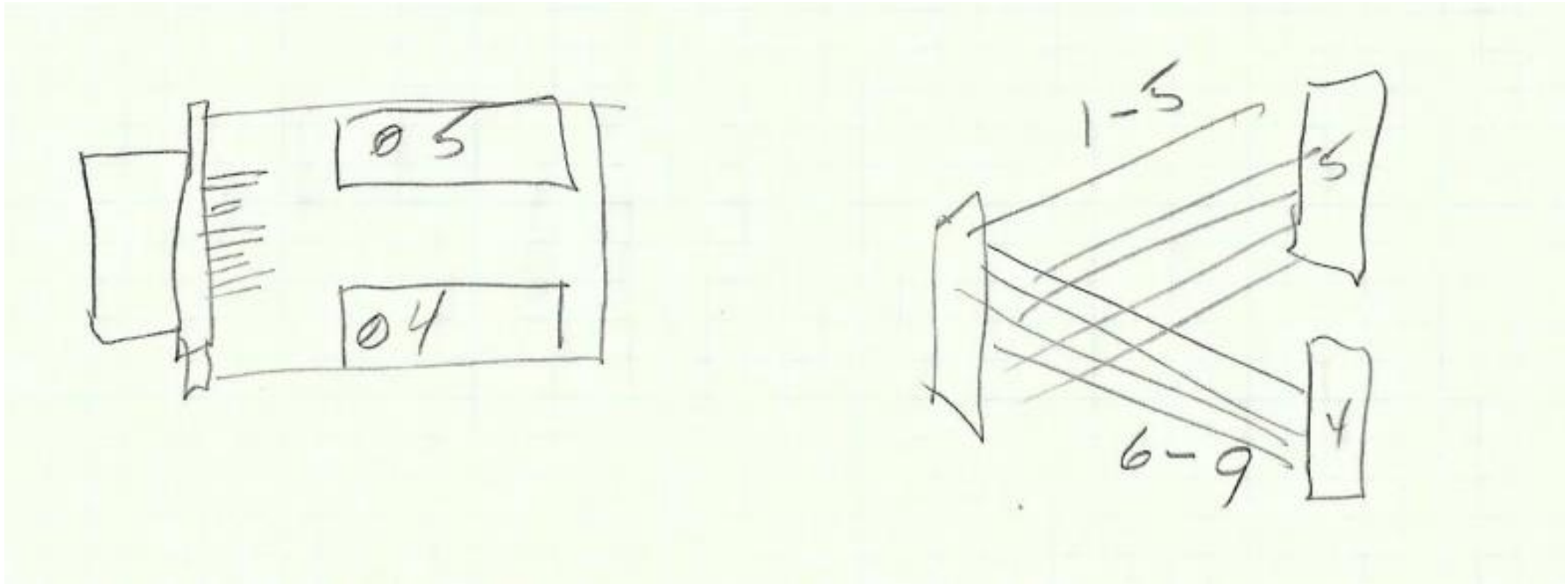


- Place Descriptive Text on the Schematic Sheet
- Place Text Box on the Schematic Sheet
- Draw Graphic Rectangle on the Schematic Sheet
- Draw Graphic Circle on the Schematic Sheet
- Draw Graphic Arc on the Schematic Sheet
- Draw Graphic Line on the Schematic Sheet
- Place a Picture on the Schematic Sheet
- Delete an Item

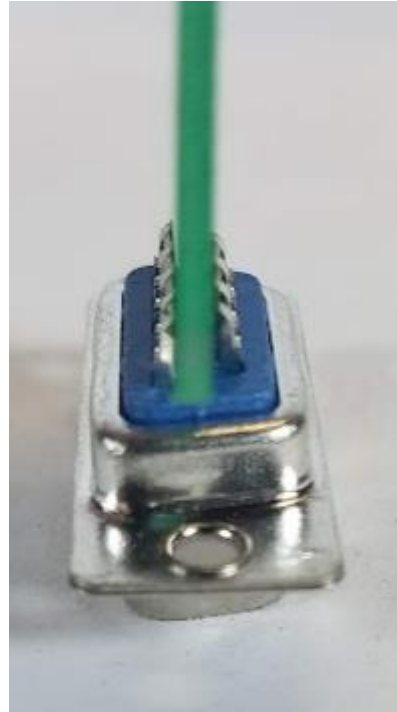
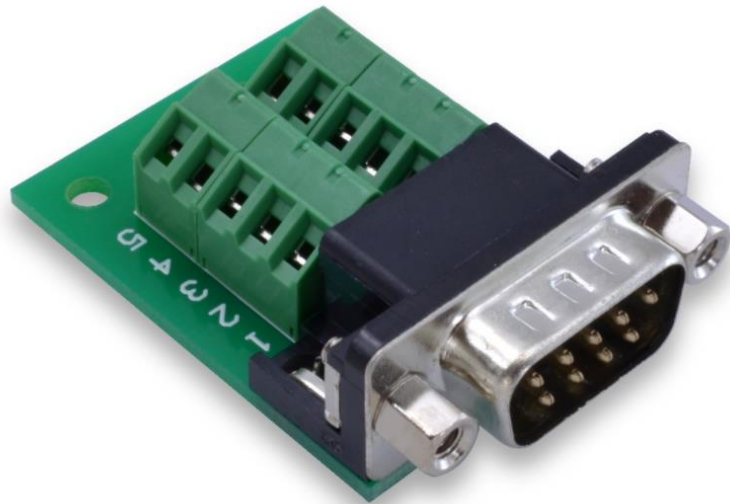
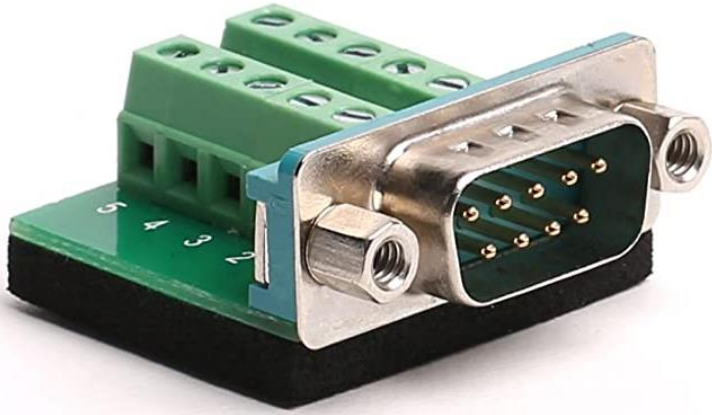
KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

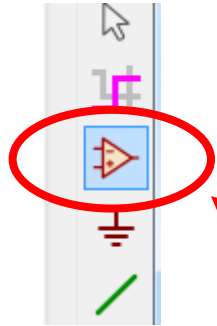
Example Project – Hand Sketch



Example Project – Parts

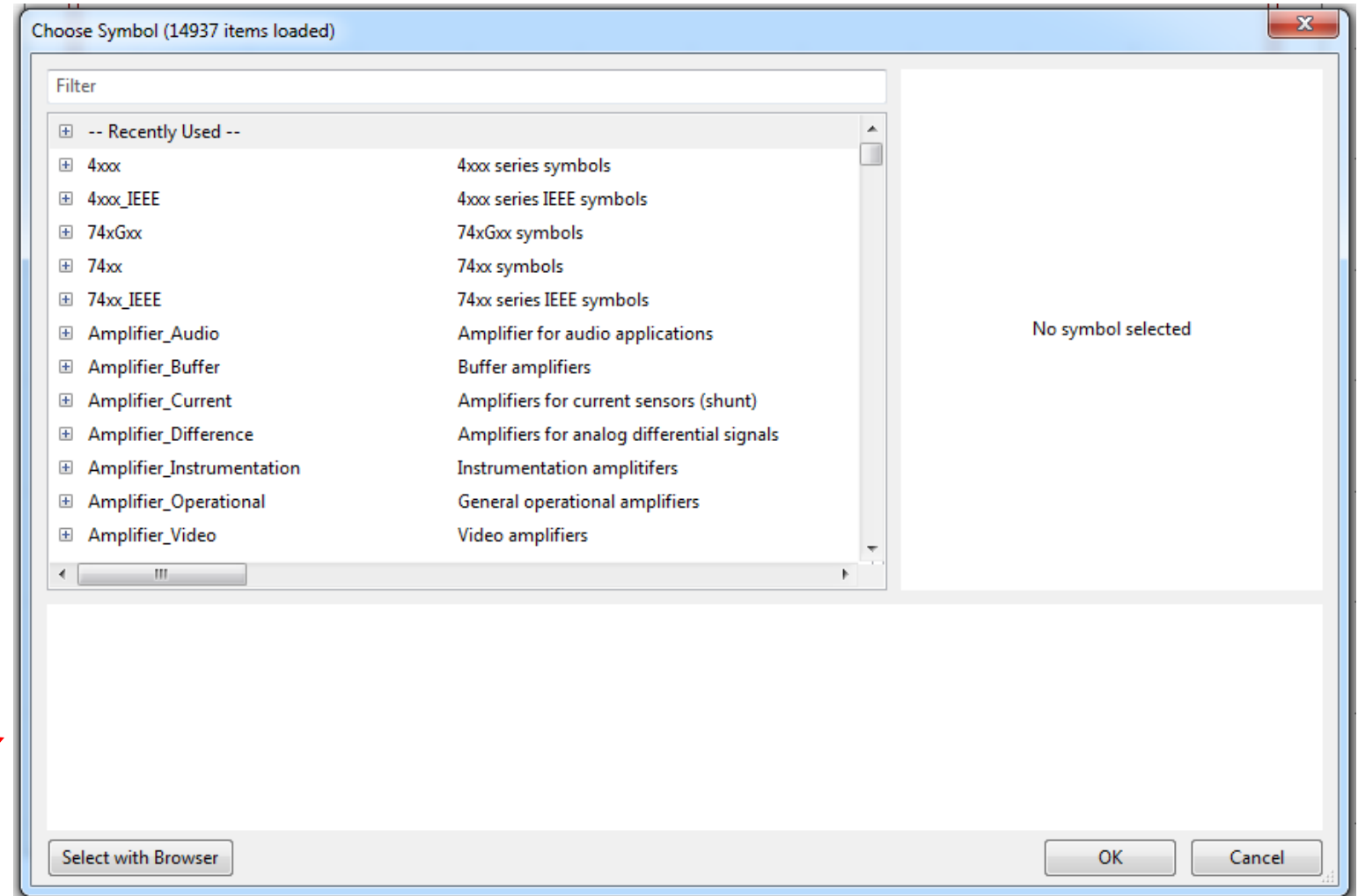


Schematic Layout Editor



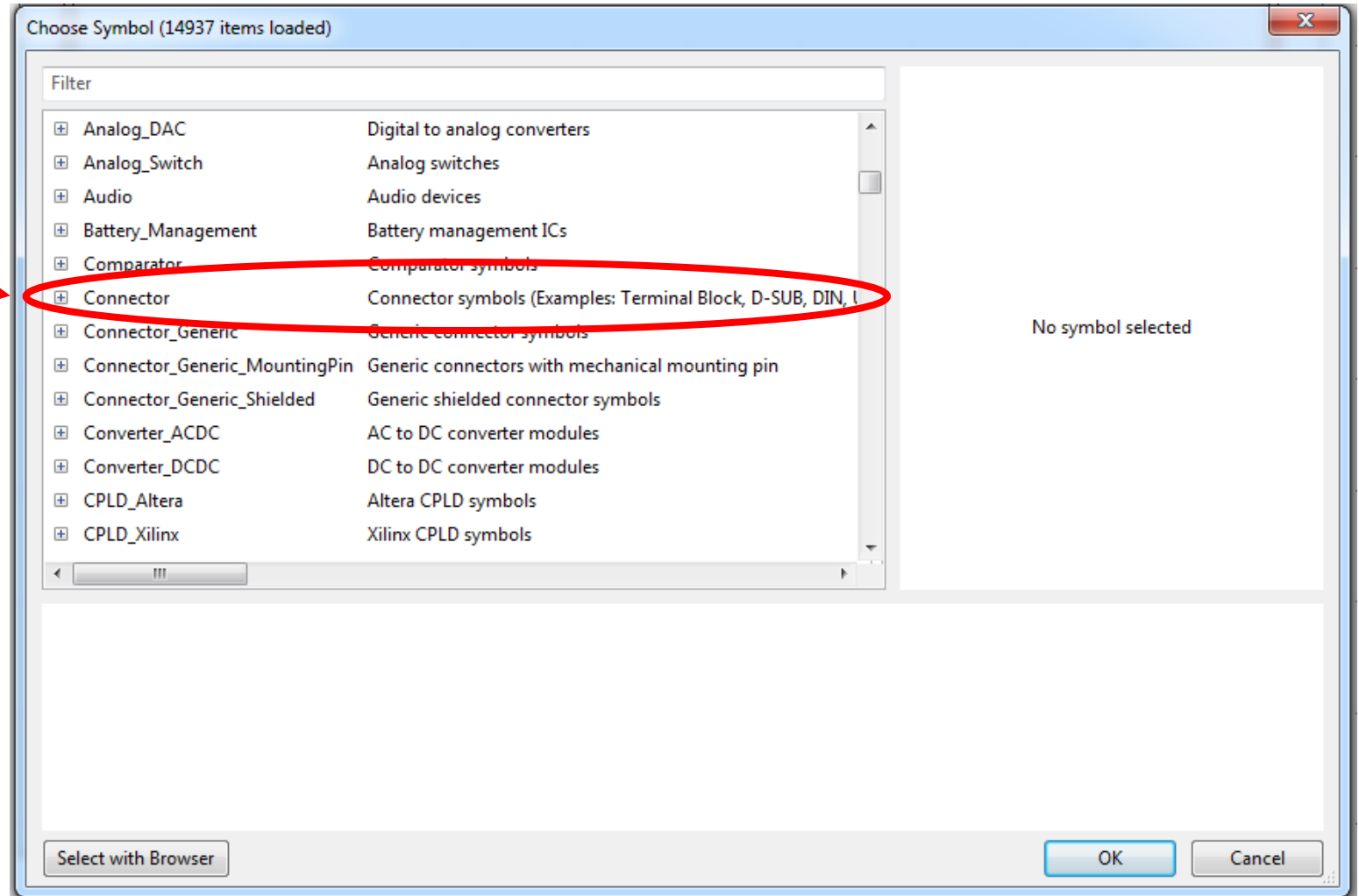
Click Place Symbol
then Click Sheet

Symbol Library
selection dialog



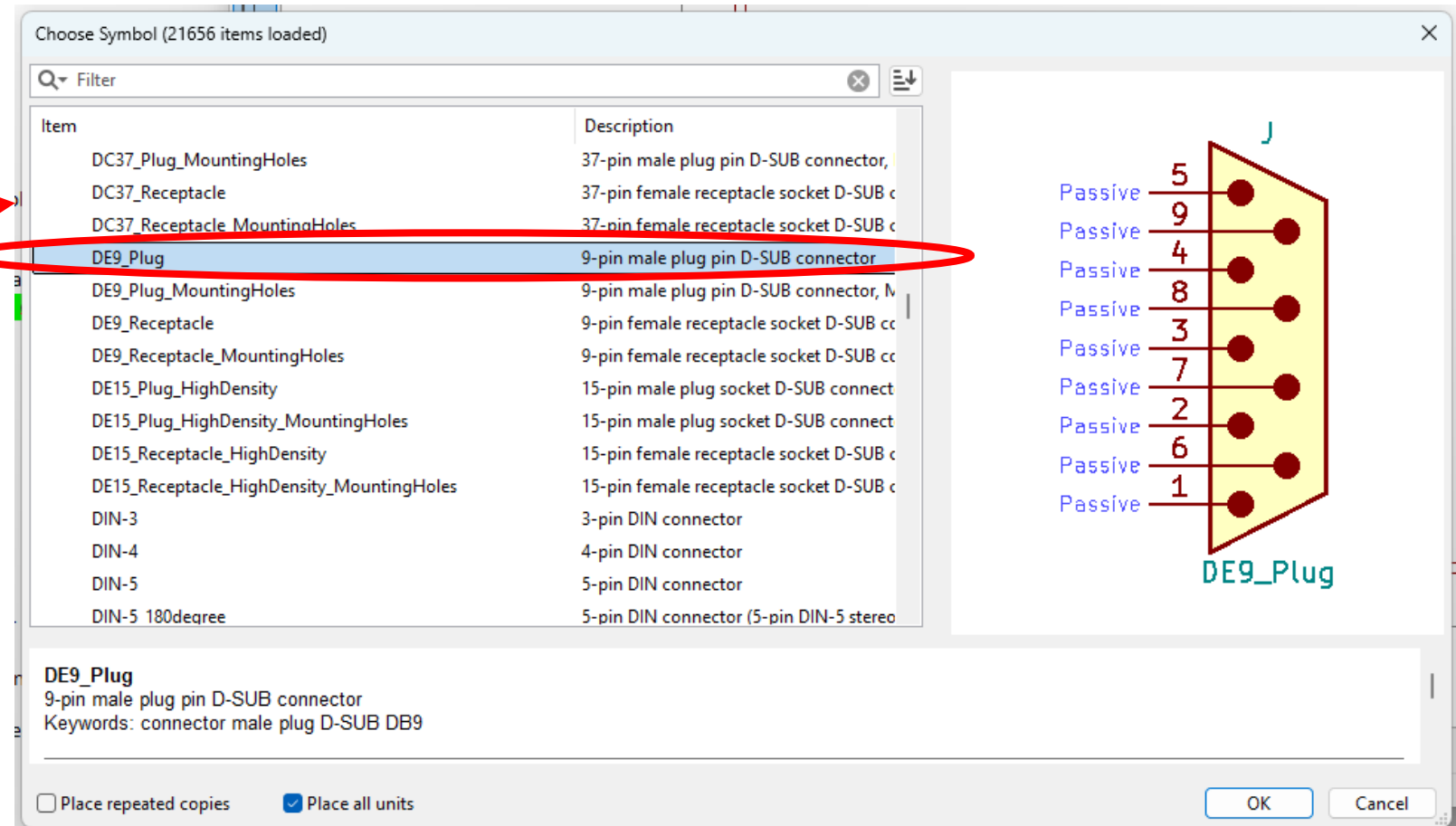
Schematic Layout Editor

Scroll to Connector
Category



Schematic Layout Editor

Scroll to DE9_Plug
Connector

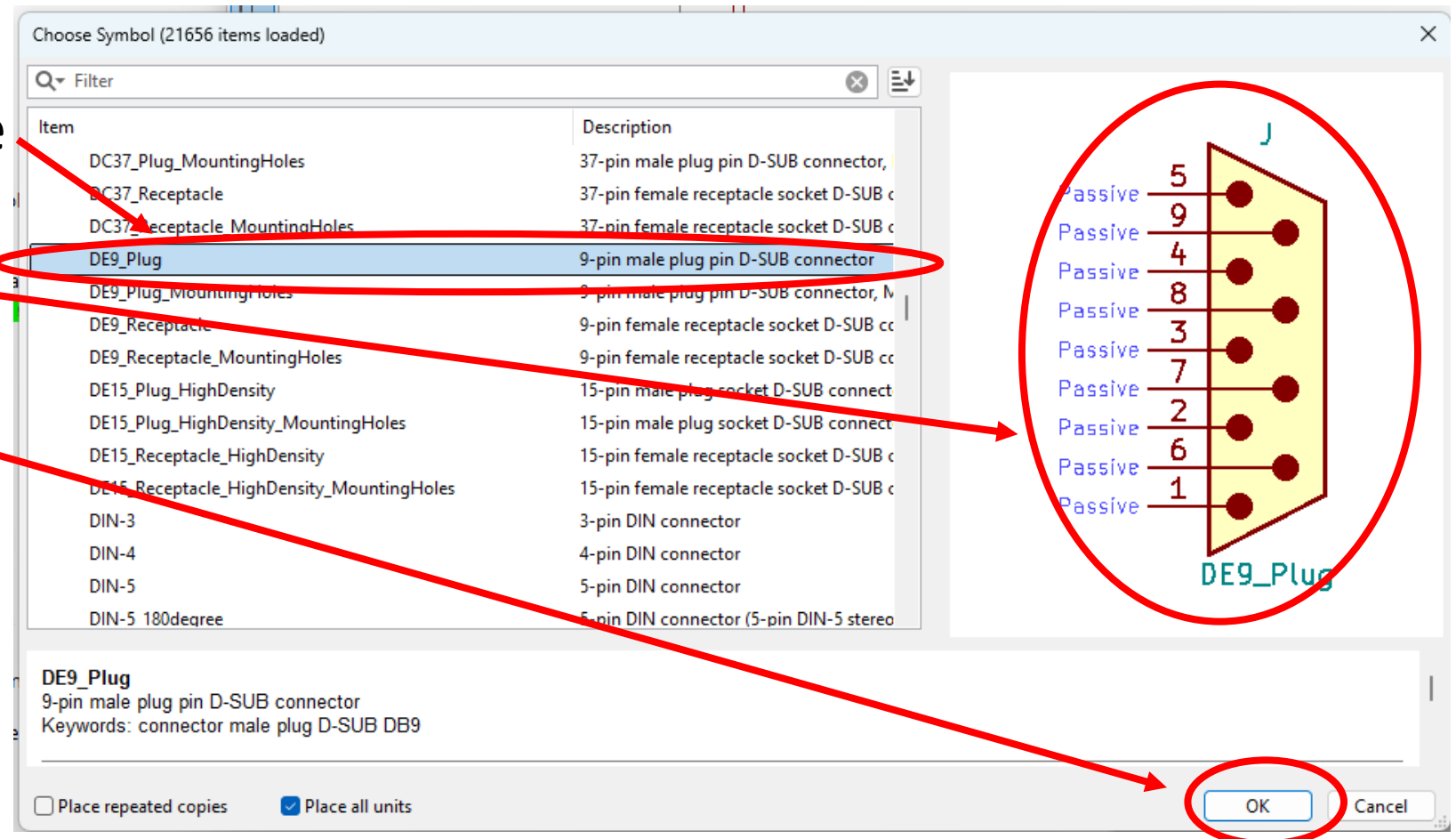


Schematic Layout Editor

Click Symbol Name

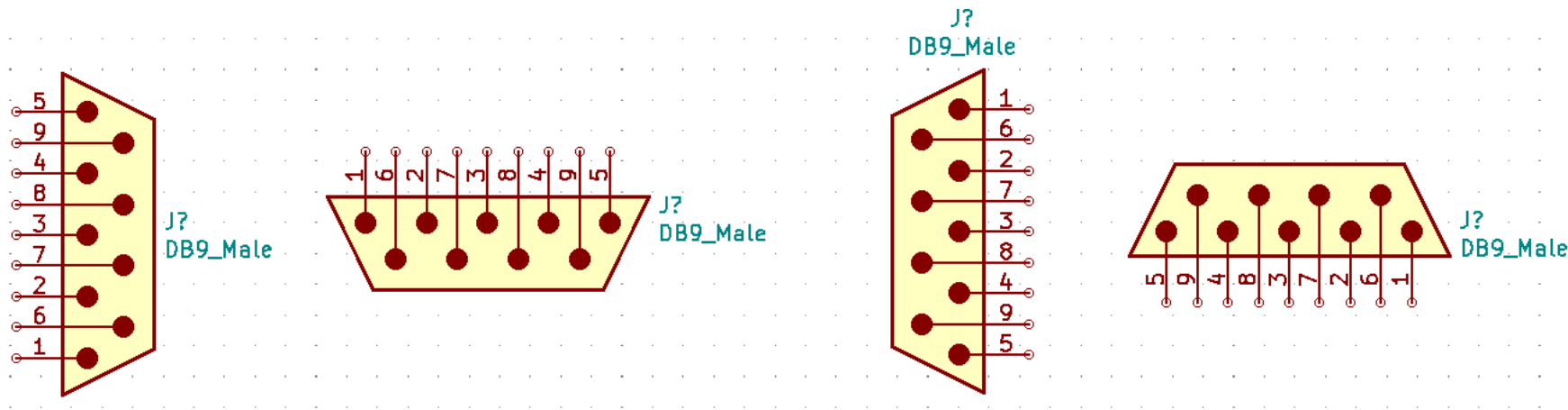
Review Symbol

Click OK to Select



Schematic Layout Editor

Press R to Rotate the Symbol to the Position You Need on the Sheet. Click to Place.



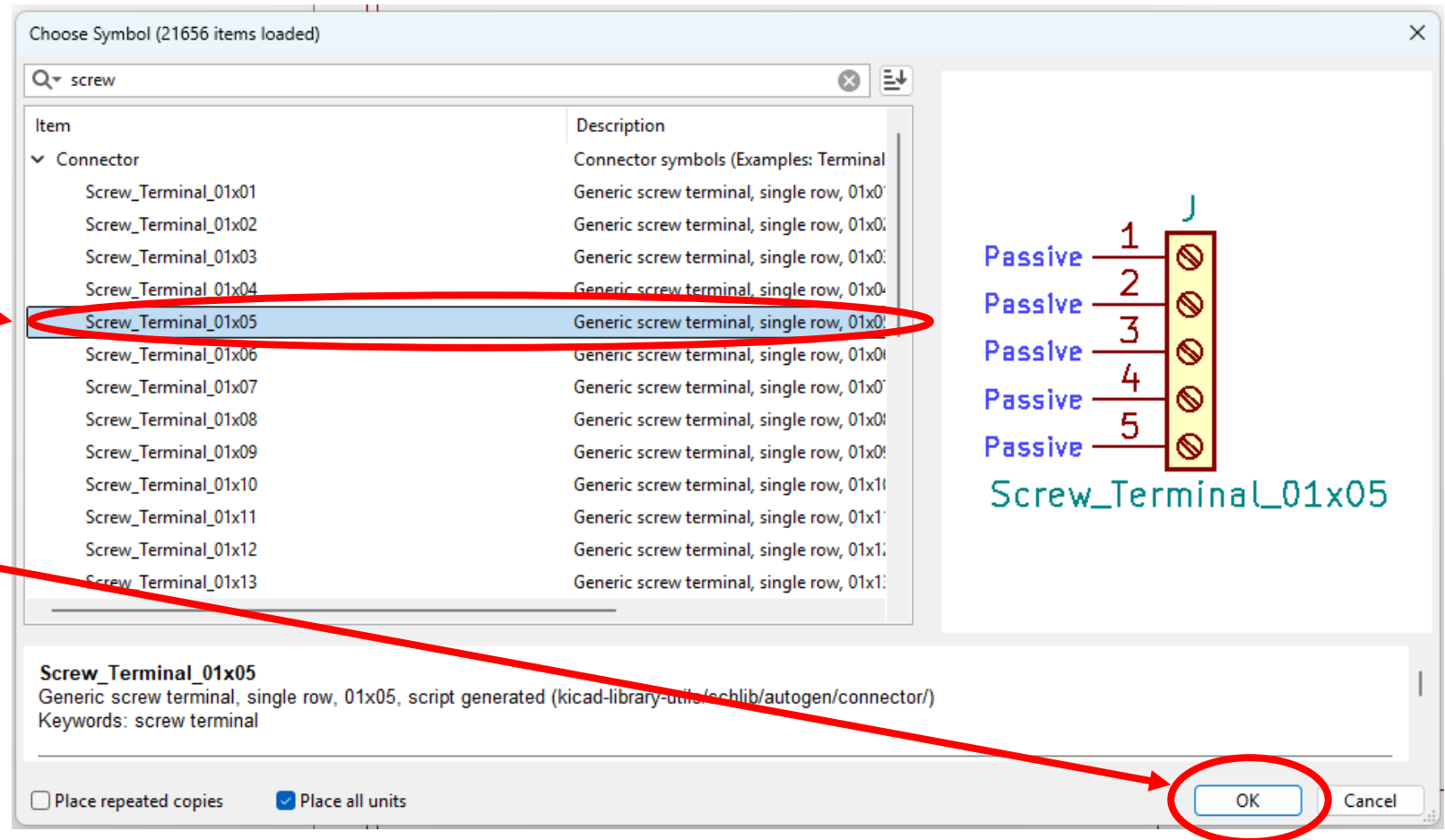
Schematic Layout Editor

Click Sheet

Scroll to
Screw_Terminal_01x05

Click OK

Click to Place on Sheet



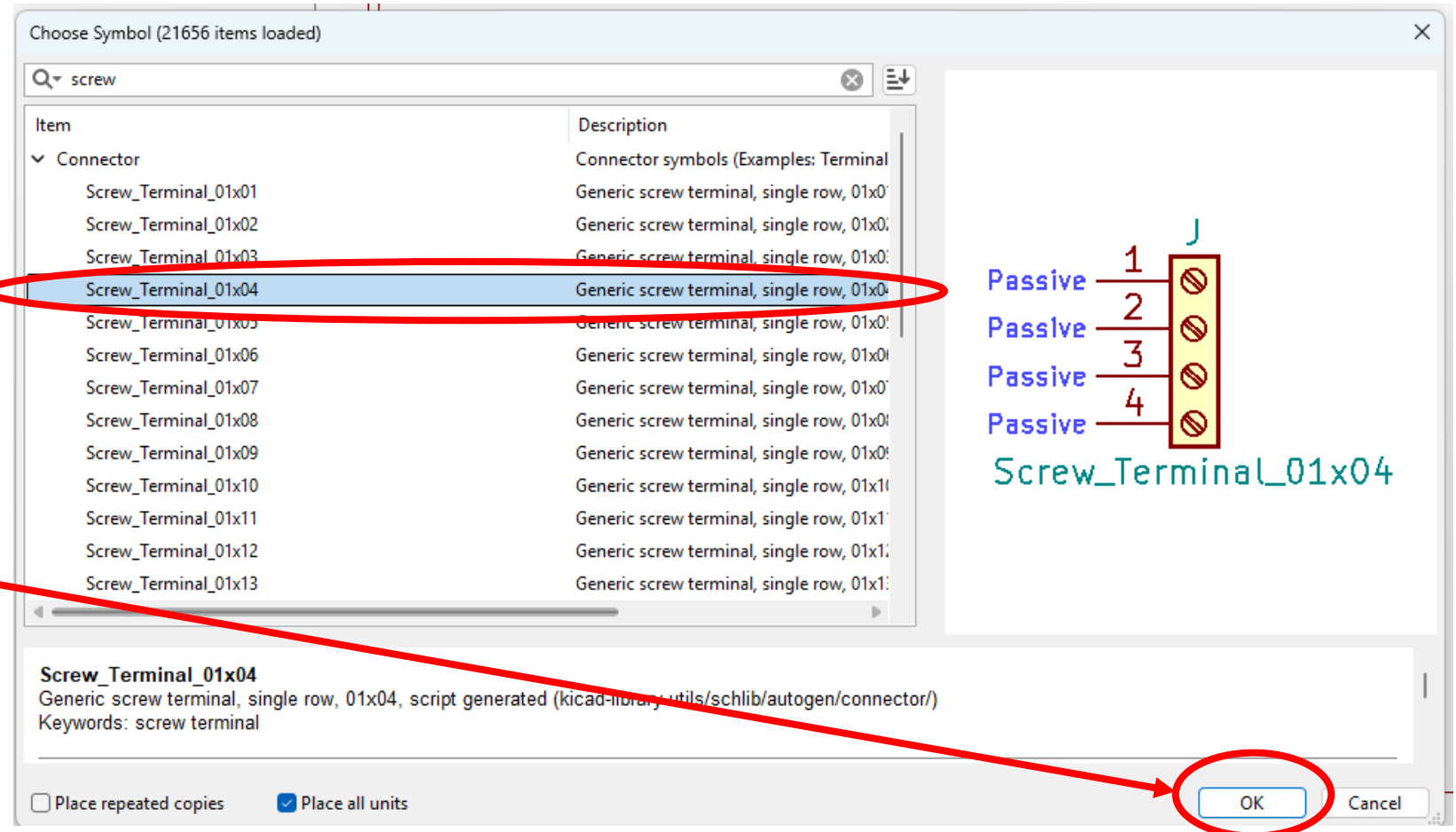
Schematic Layout Editor

Click Sheet

Scroll to
Screw_Terminal_01x04

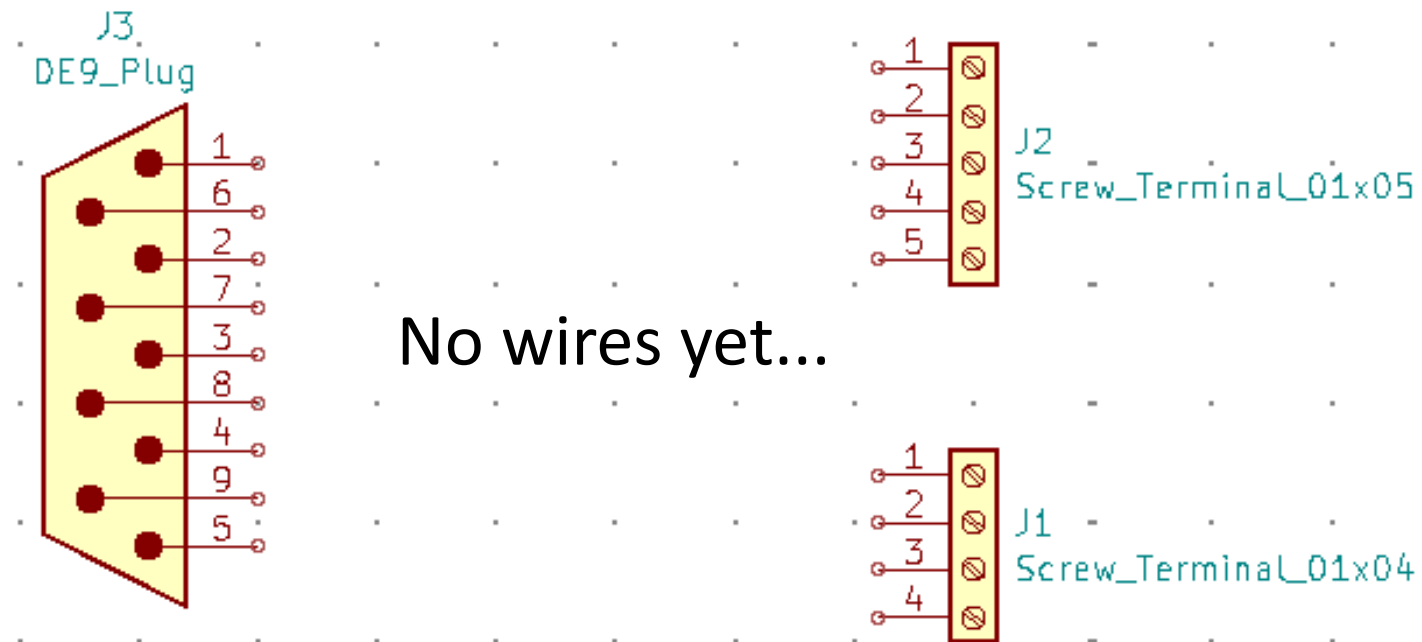
Click OK

Click to Place on Sheet



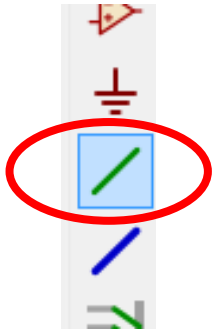
Schematic Layout Editor

Schematic So Far...

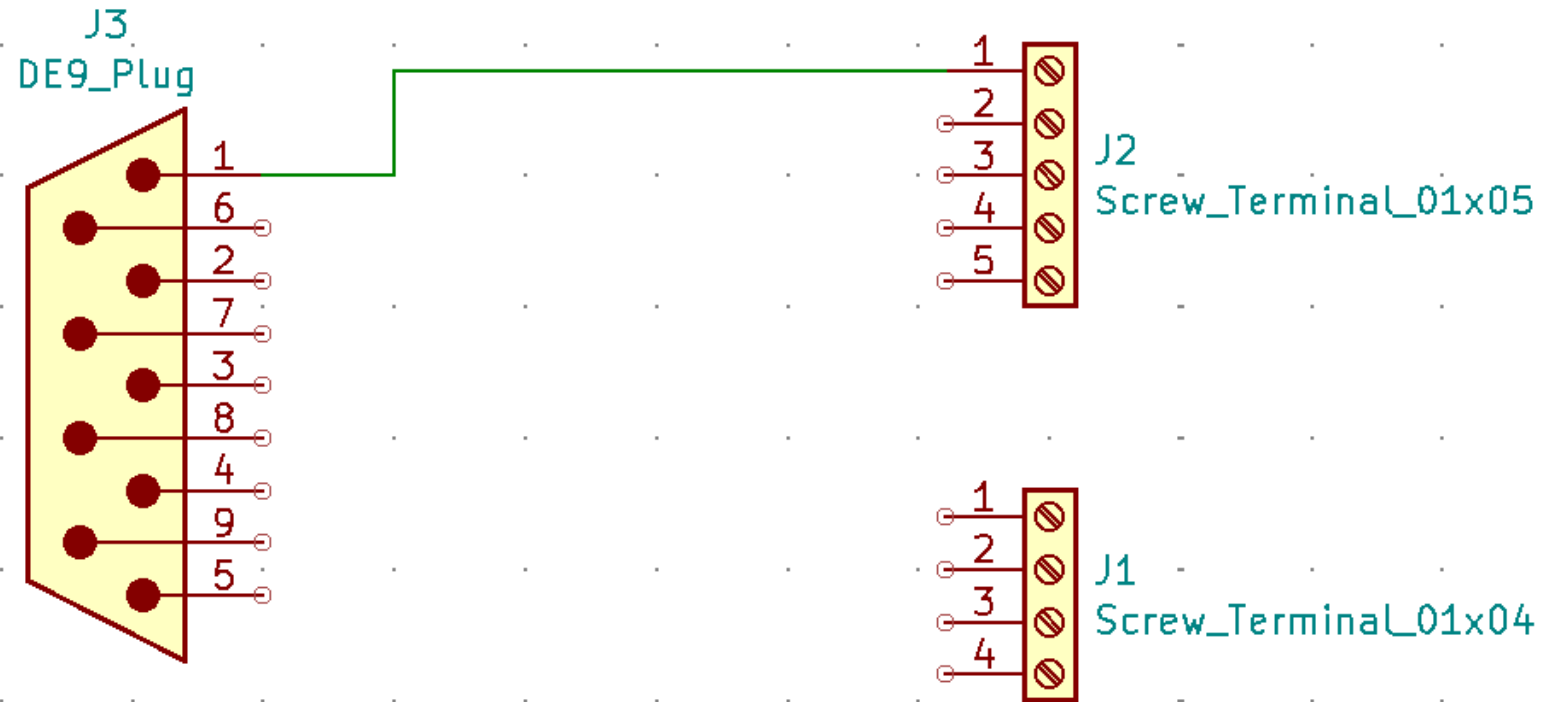


Schematic Layout Editor

Click Place Wire



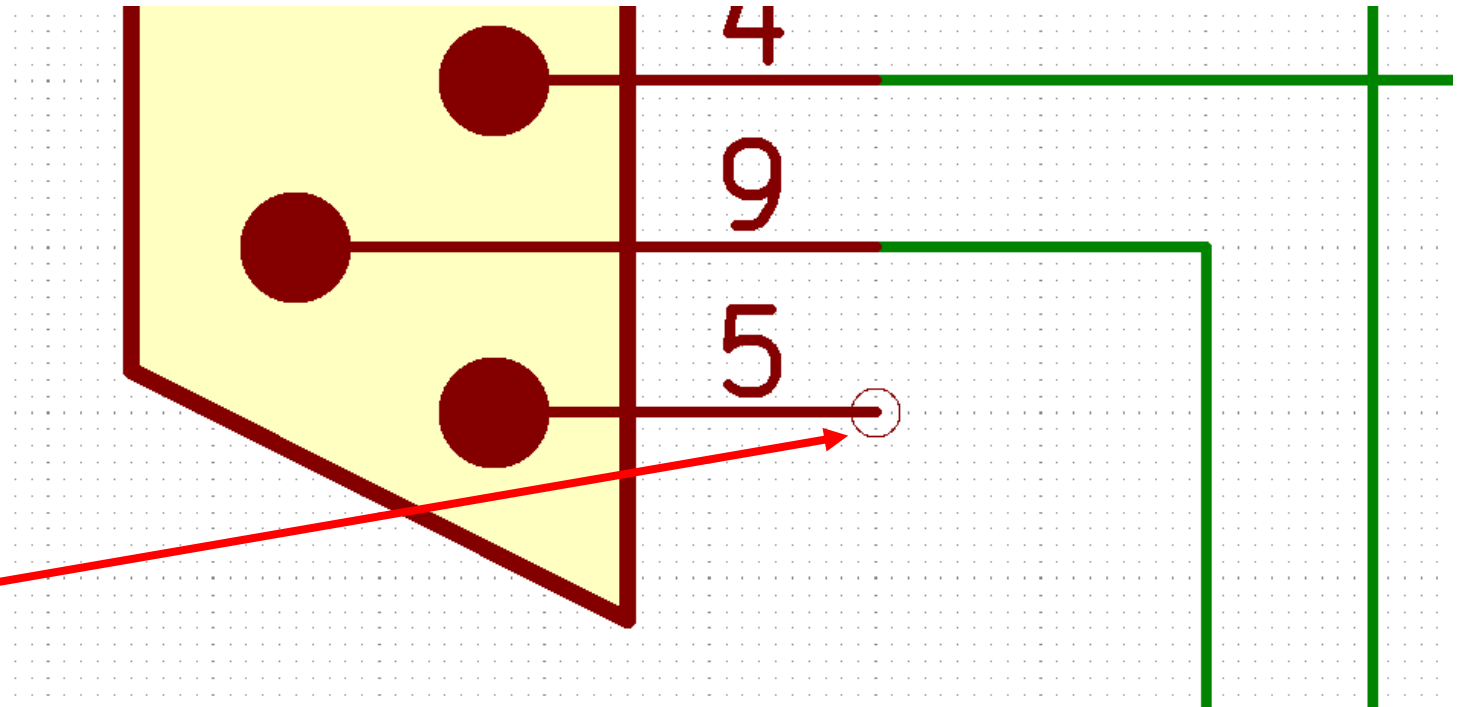
Click Start “Bubble”
Click Corner
Click End “Bubble”
On Each Wire...



Schematic Layout Editor

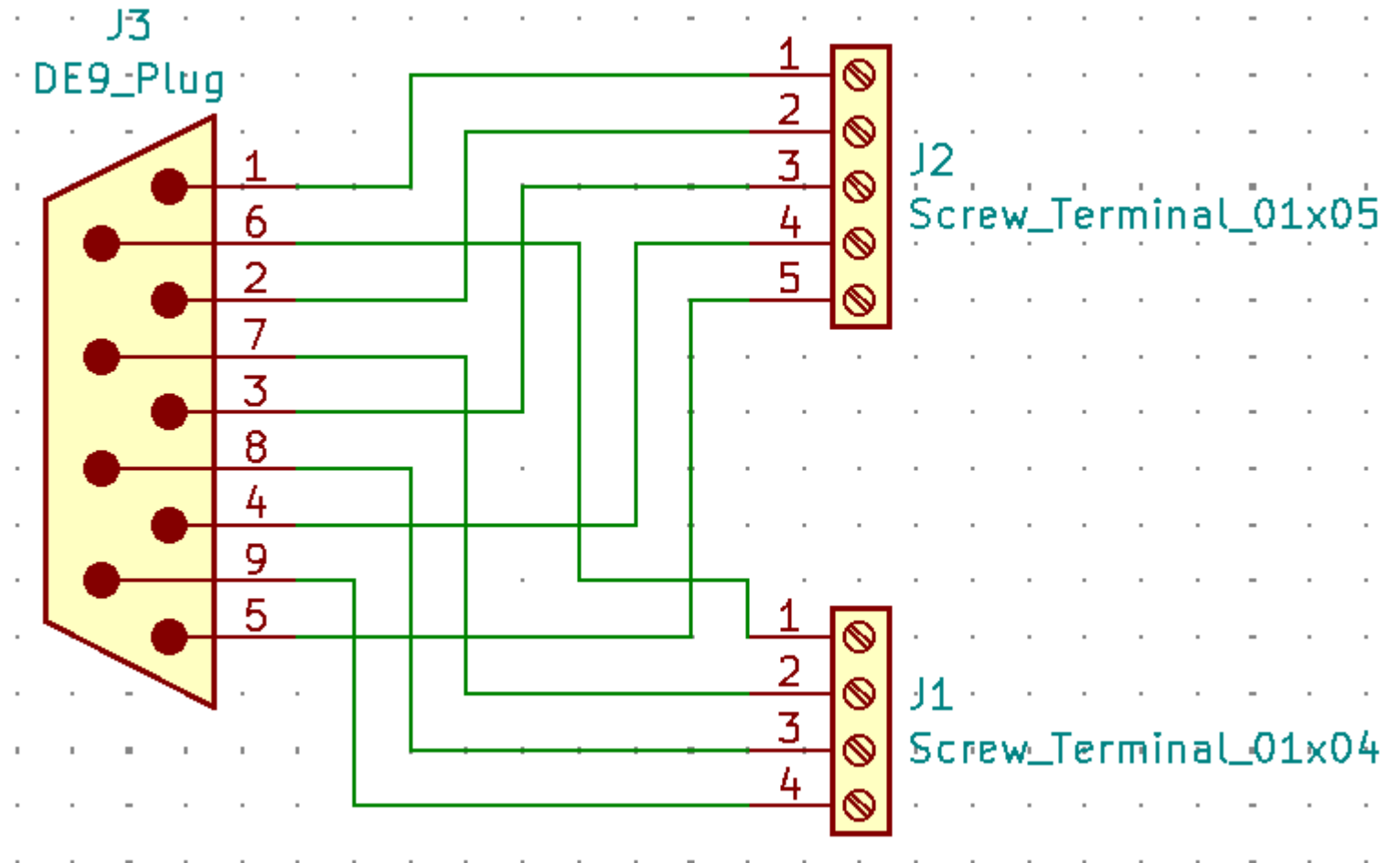
Zoom In for Detail
on “Bubbles”

Click in the Middle
of the “Bubble”



Schematic Layout Editor

Schematic Done!

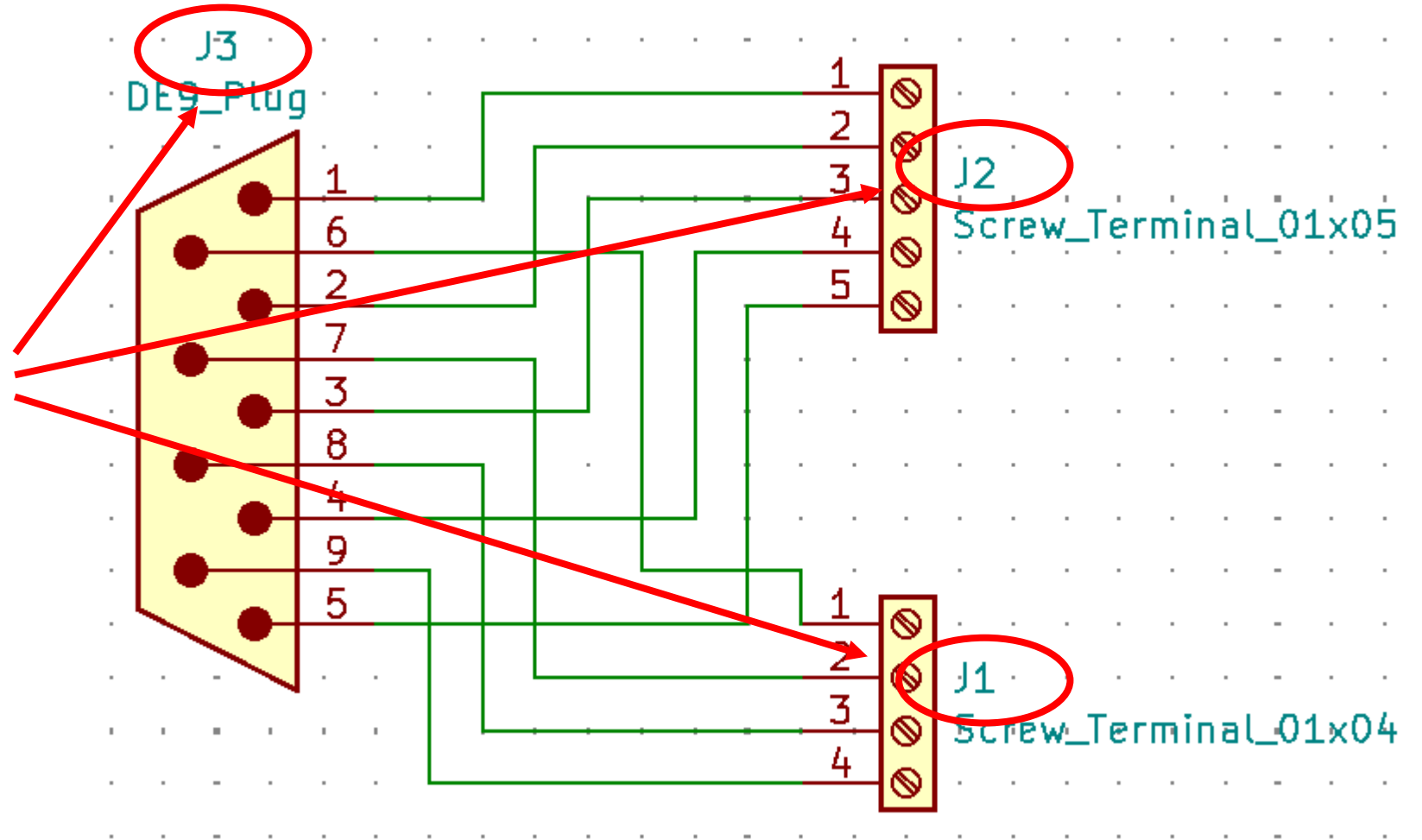


KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

Schematic Layout Editor

Could Annotate

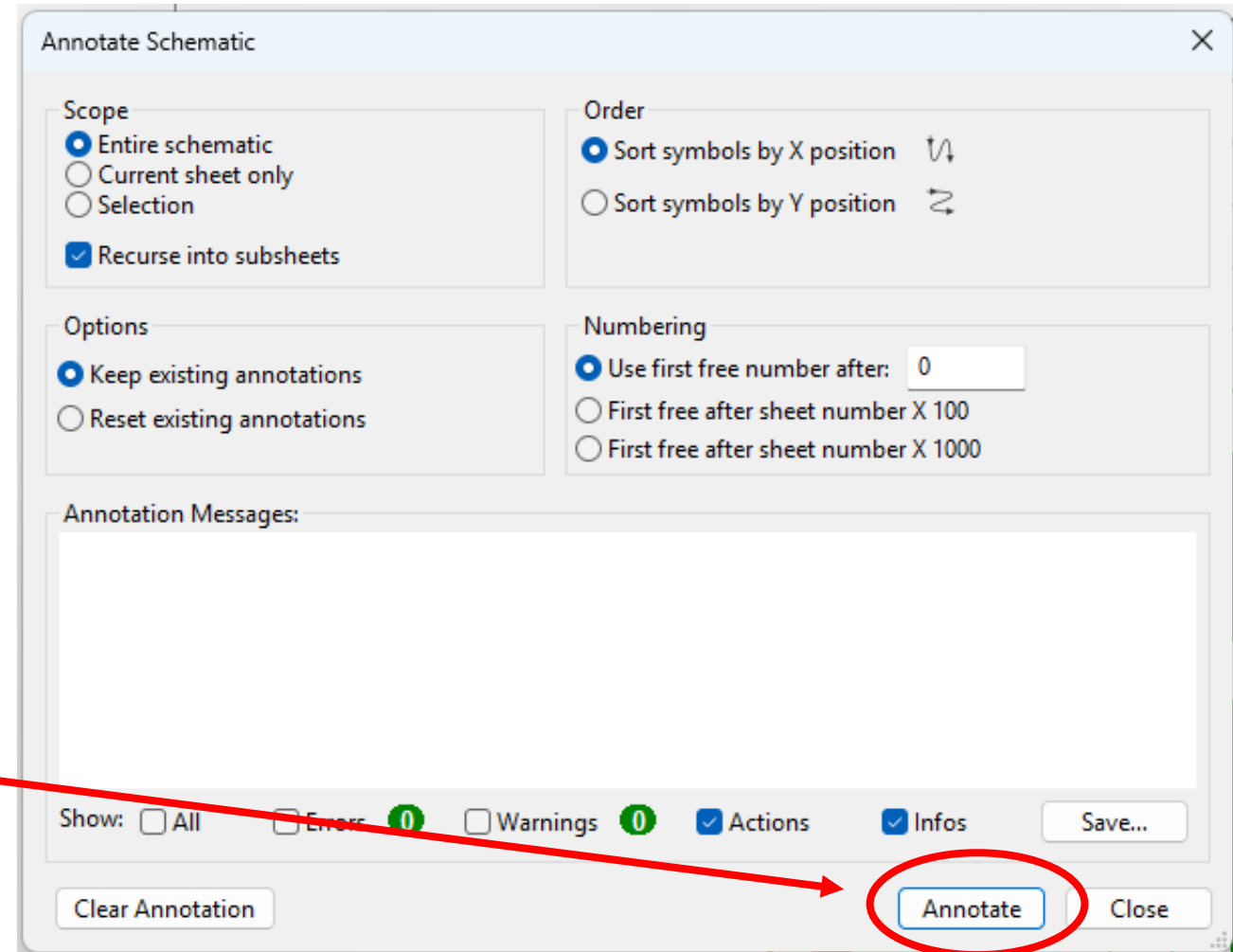


Schematic Layout Editor

Click Annotate Icon



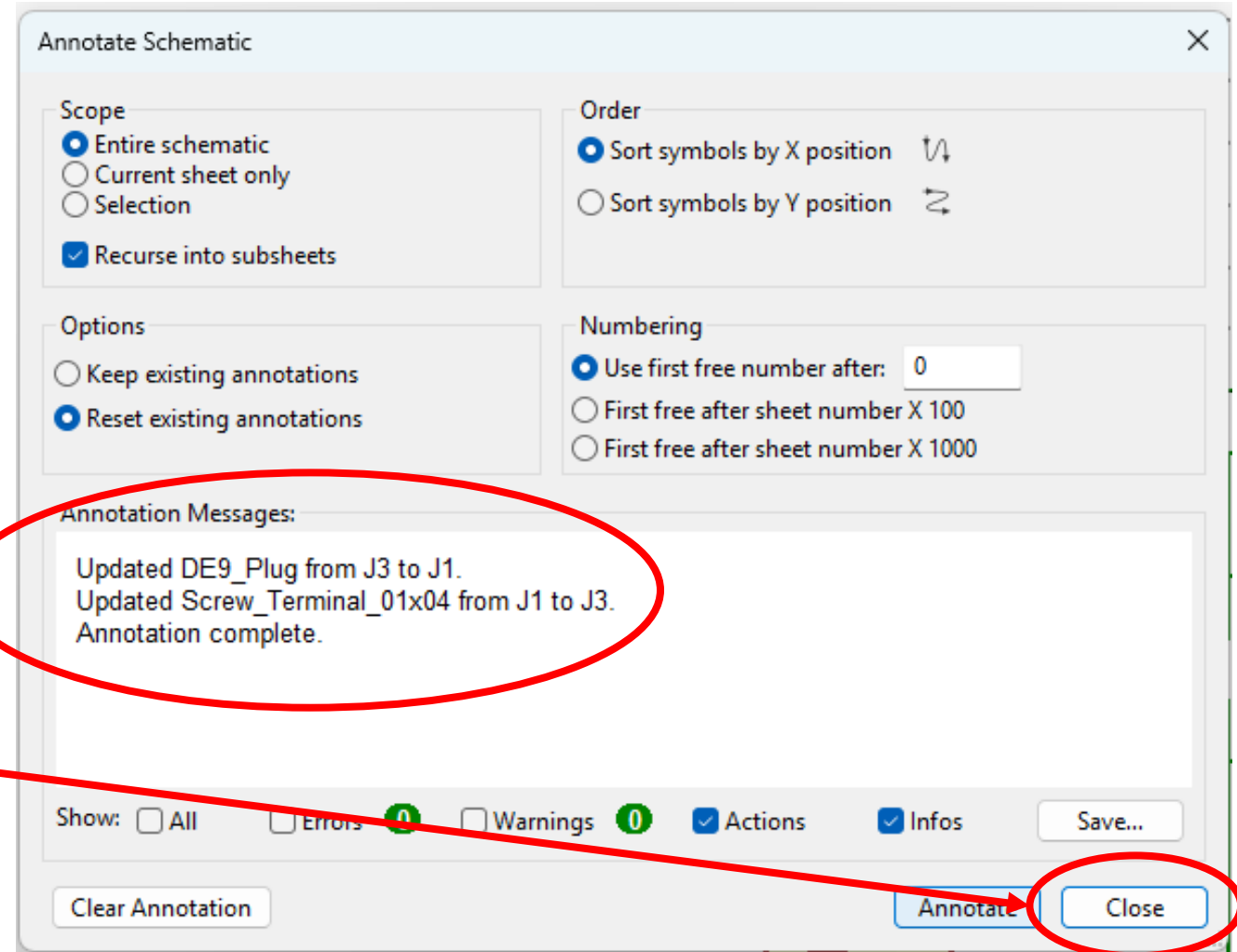
Click Annotate on the dialog



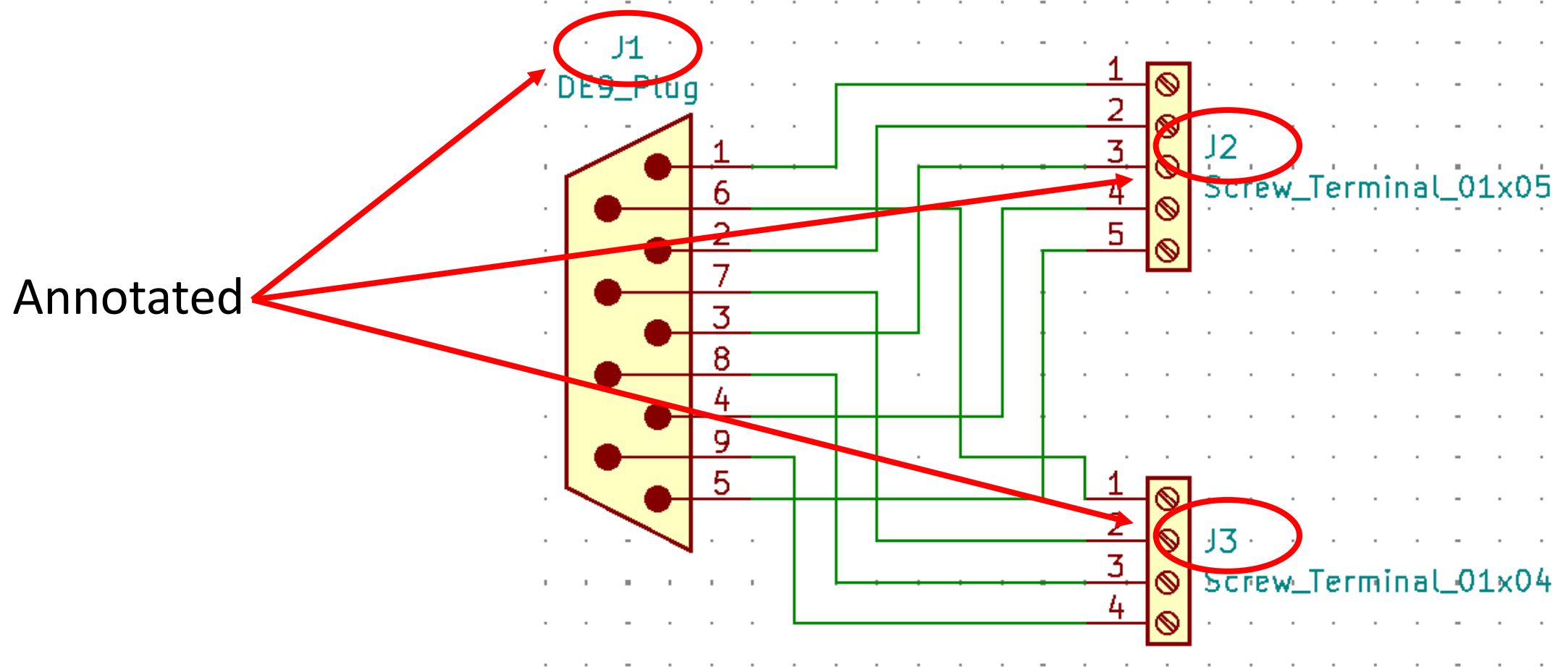
Schematic Layout Editor

Annotation Results

Click Close



Schematic Layout Editor

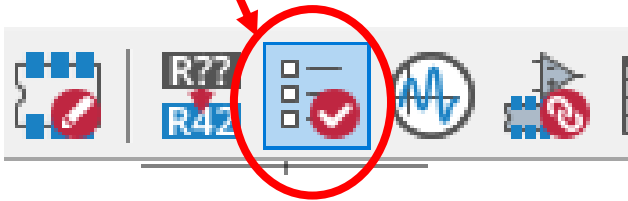


KiCad – Work Flow

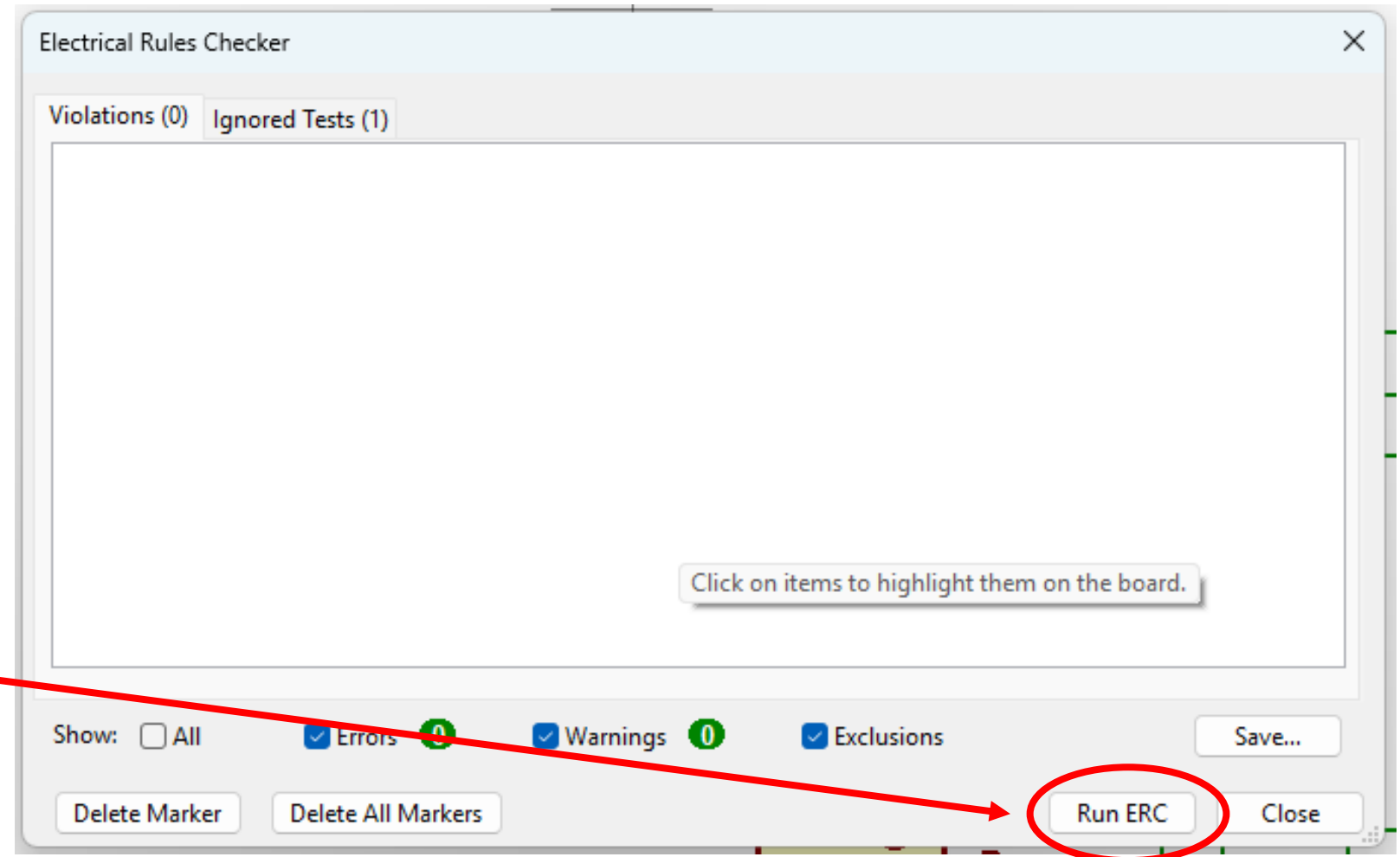
- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

Schematic Layout Editor

Click Electrical Rule Check Icon



Click Run

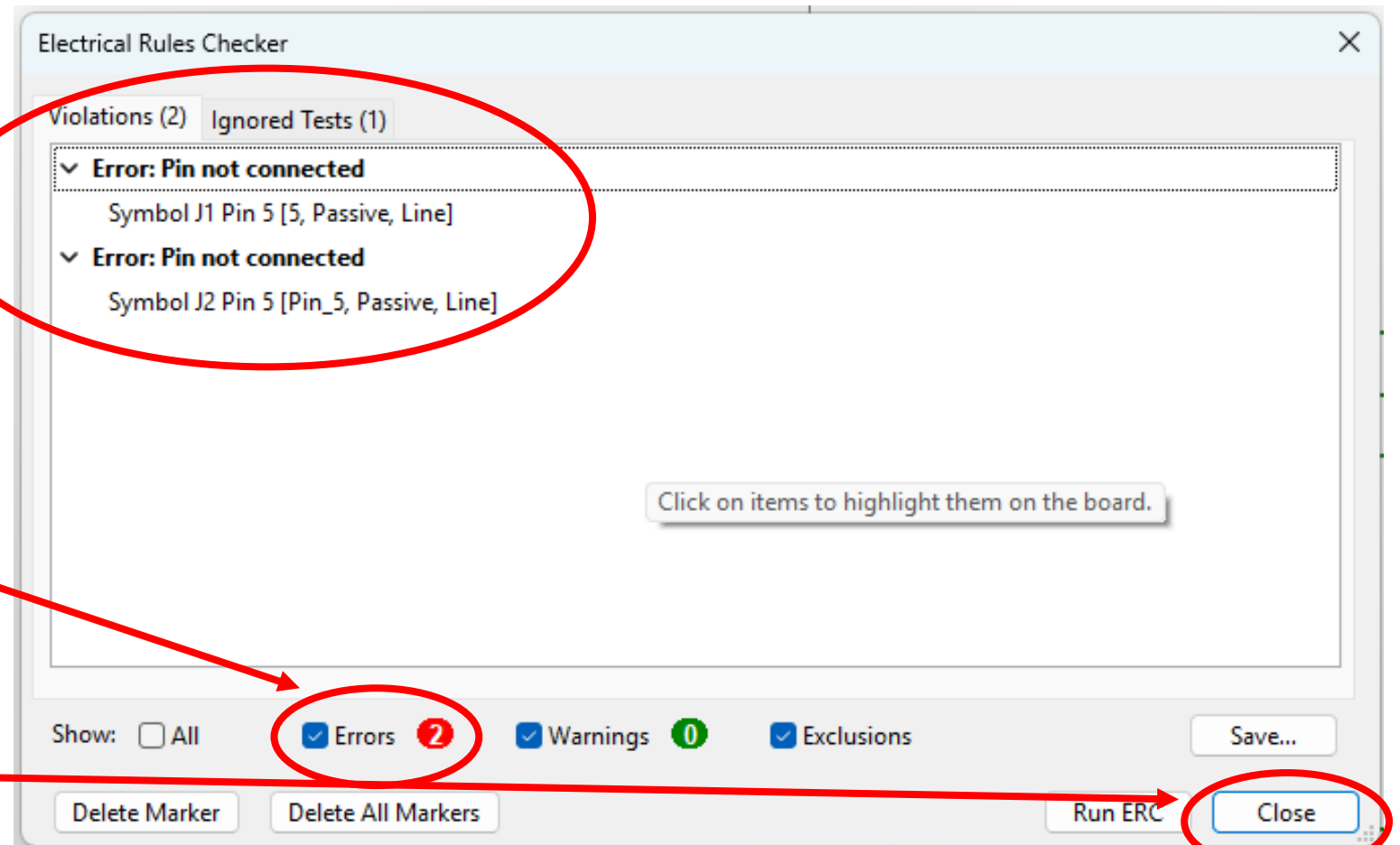


Schematic Layout Editor

Example of Running
the Electrical Rule
Check with a
Missing Connection

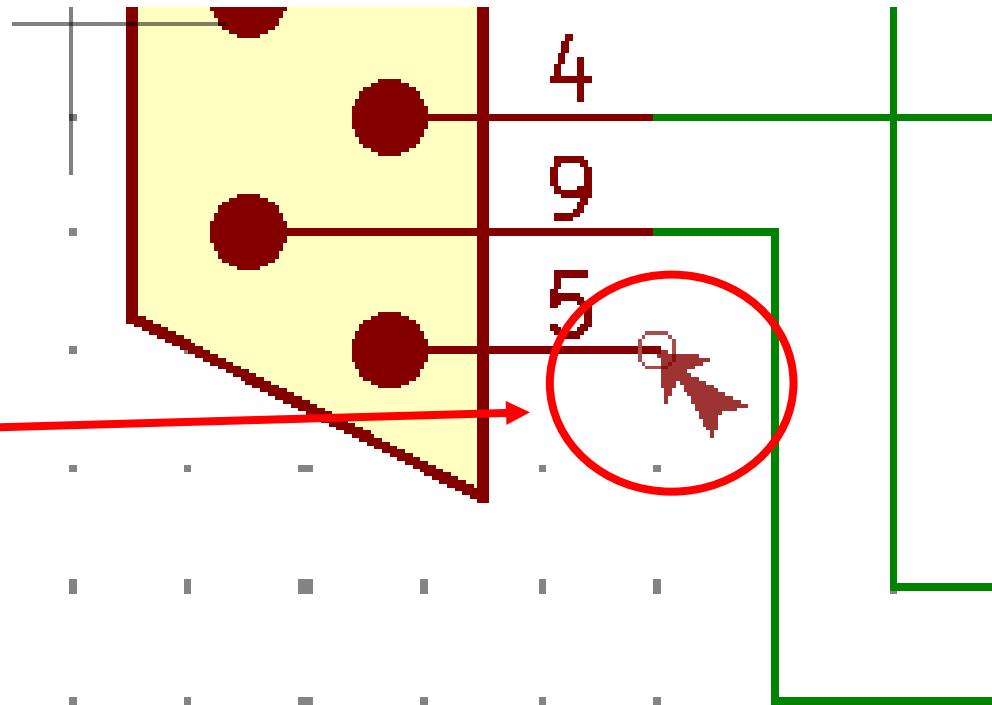
Check for Errors

Click Close



Schematic Layout Editor

Red Marker Arrows Point to Rule Check Issues

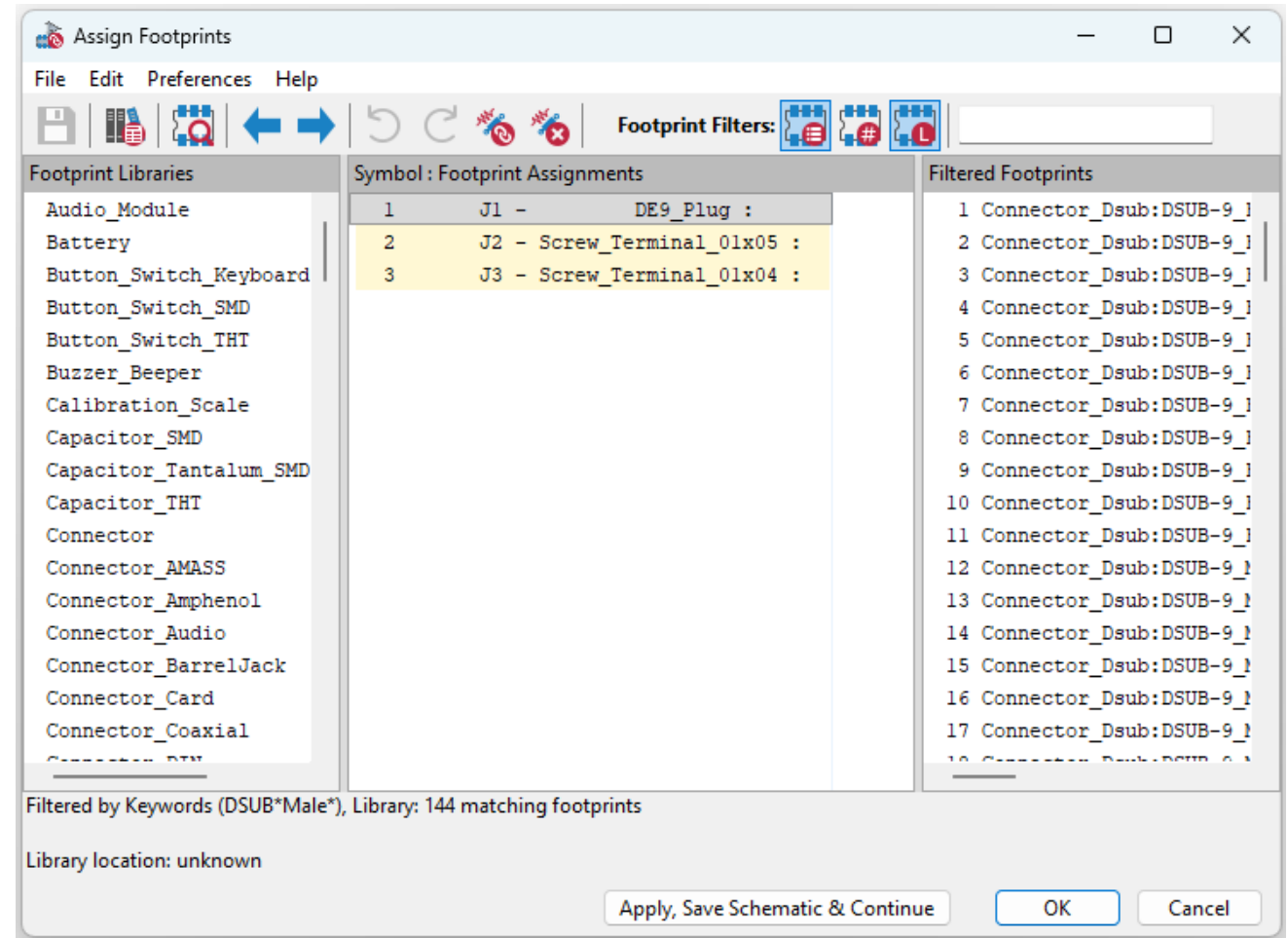


KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Import Netlist
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

Schematic Layout Editor

Click Assign Footprints
Icon

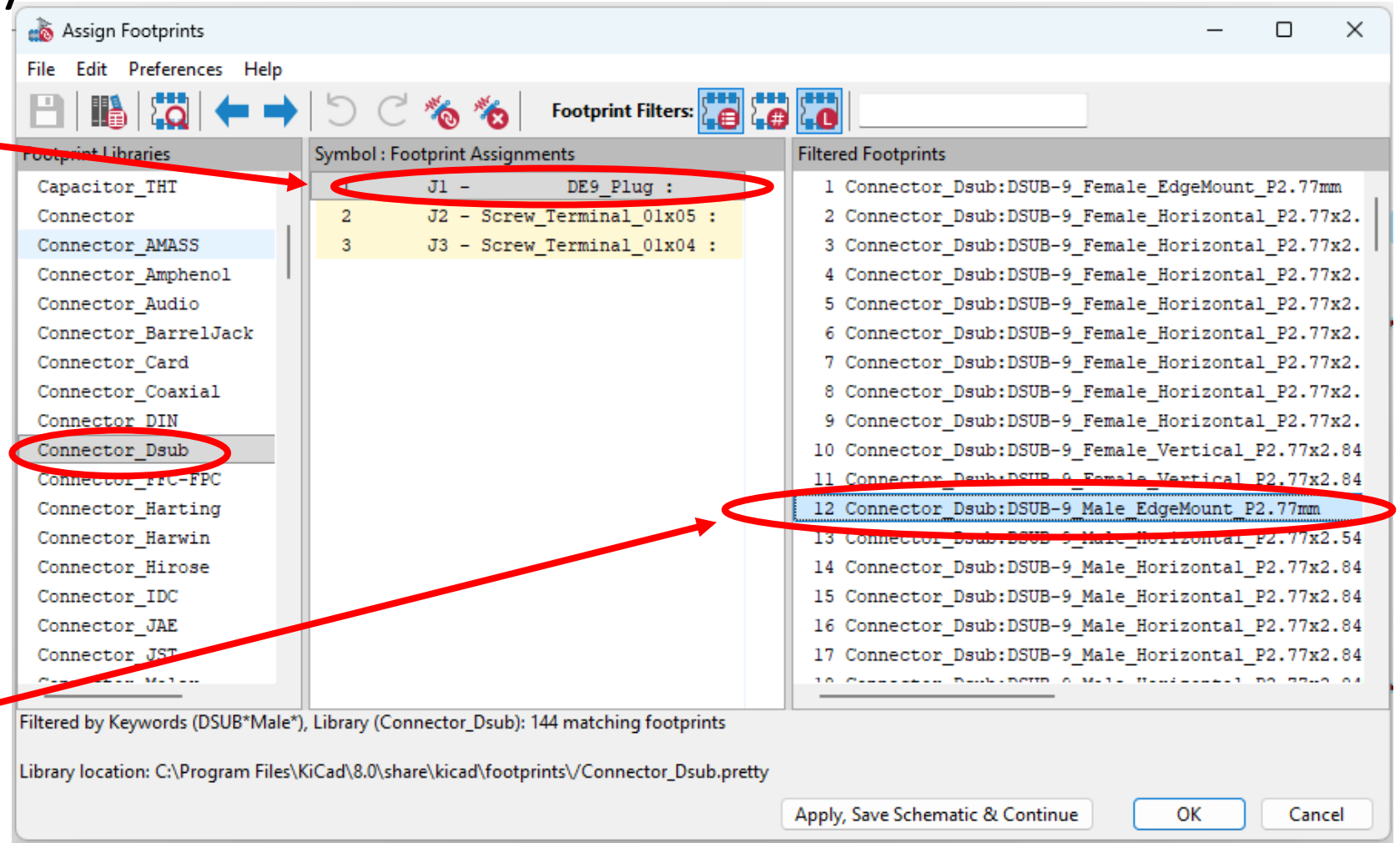


Schematic Layout Editor

Click J1

Select Library
Category

Double-Click
Footprint

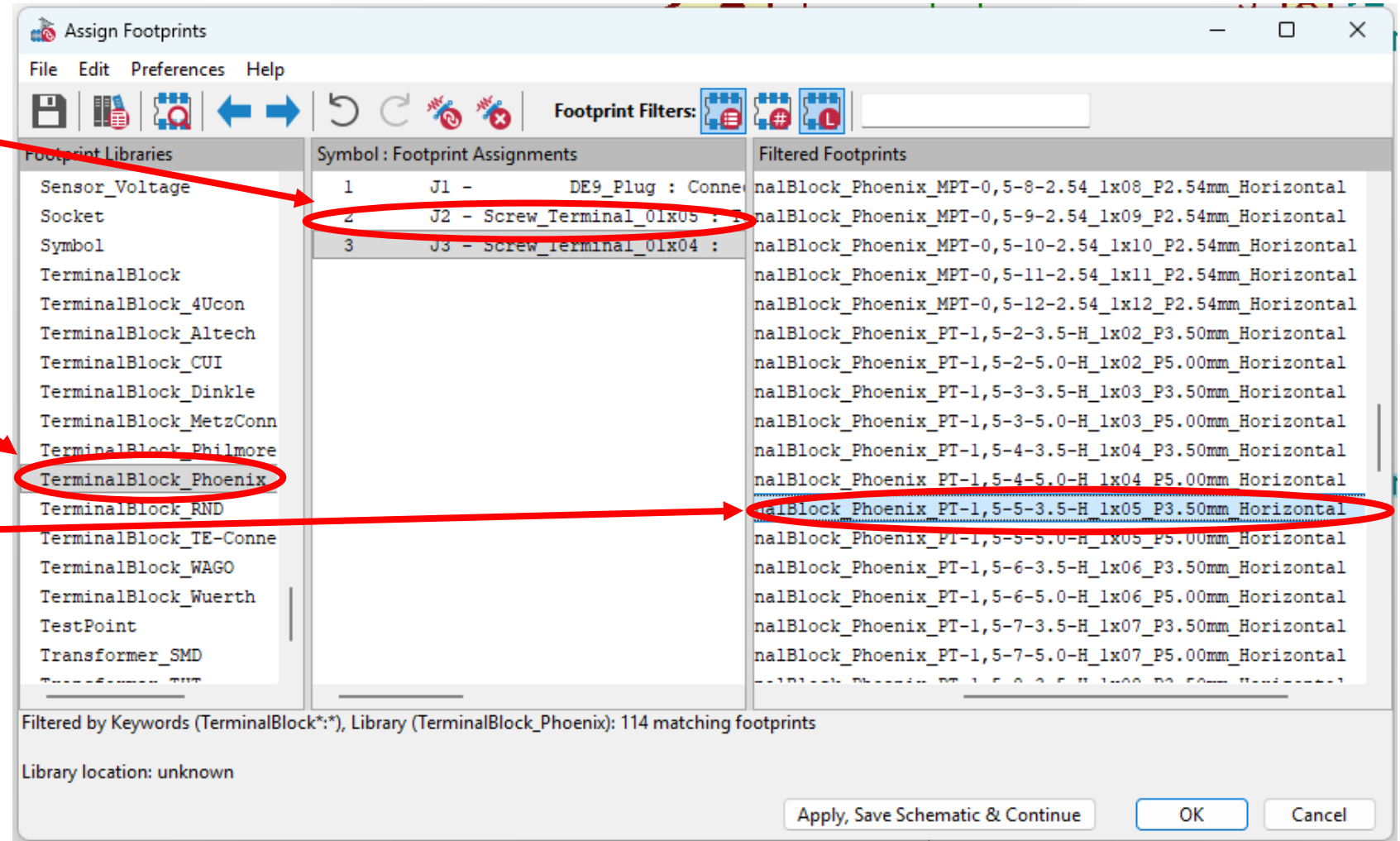


Schematic Layout Editor

Click J2

Select Library
Category

Double-Click
Footprint



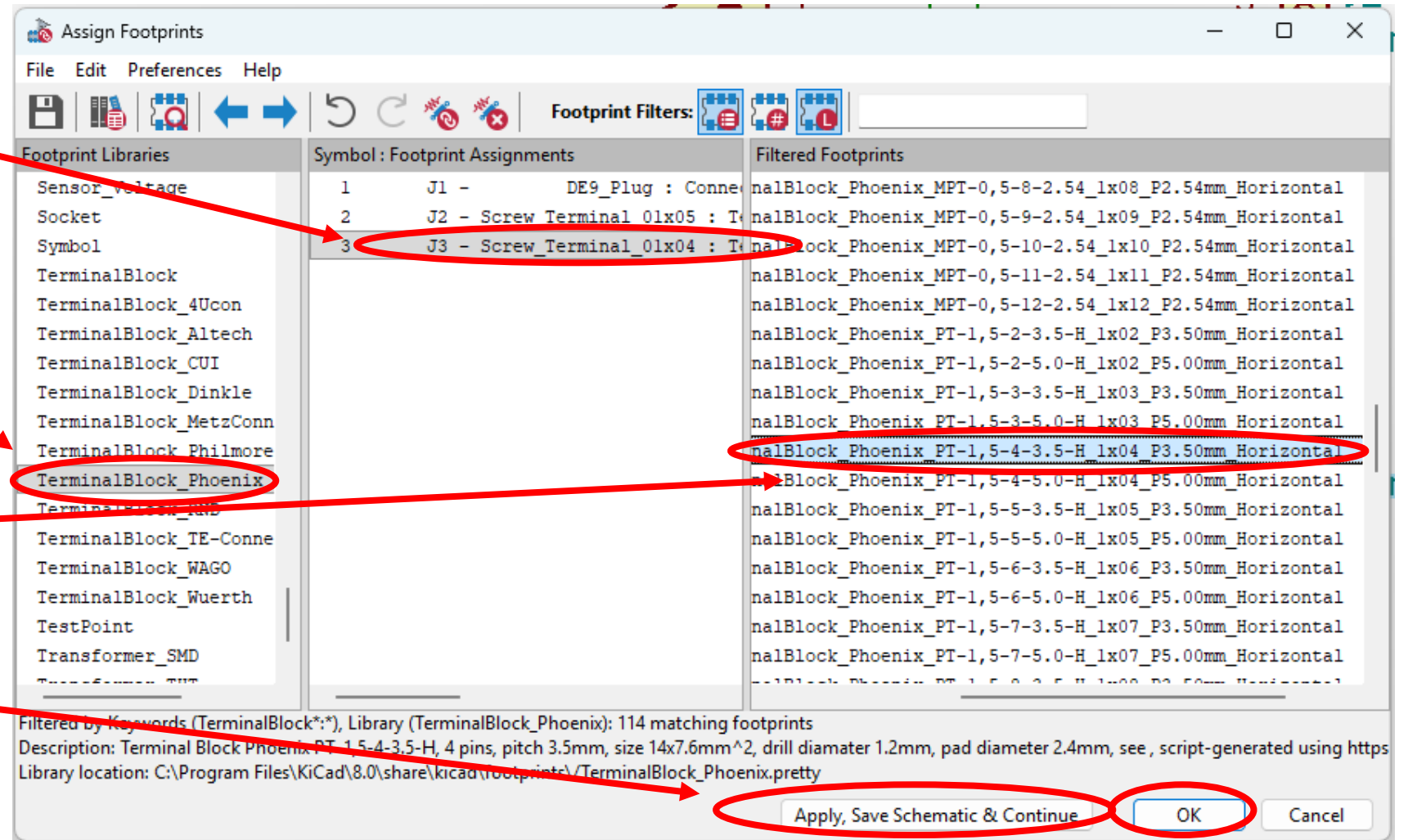
Schematic Layout Editor

Click J3

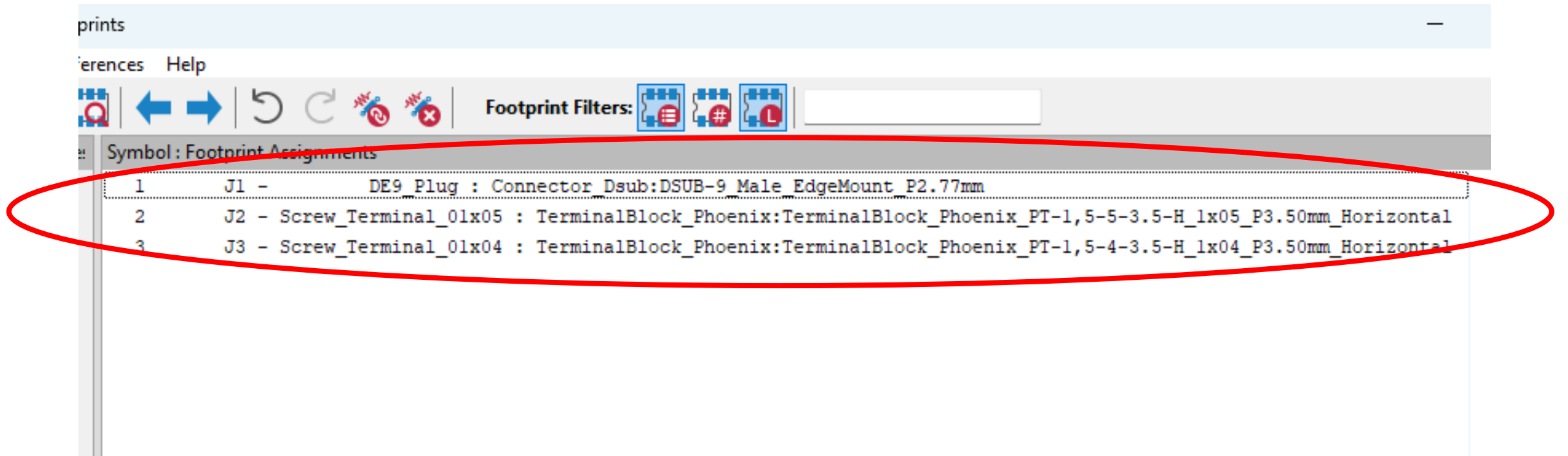
Select Library
Category

Double-Click
Footprint

Apply
then OK



Schematic Layout Editor

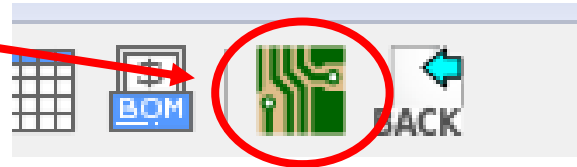


KiCad – Work Flow

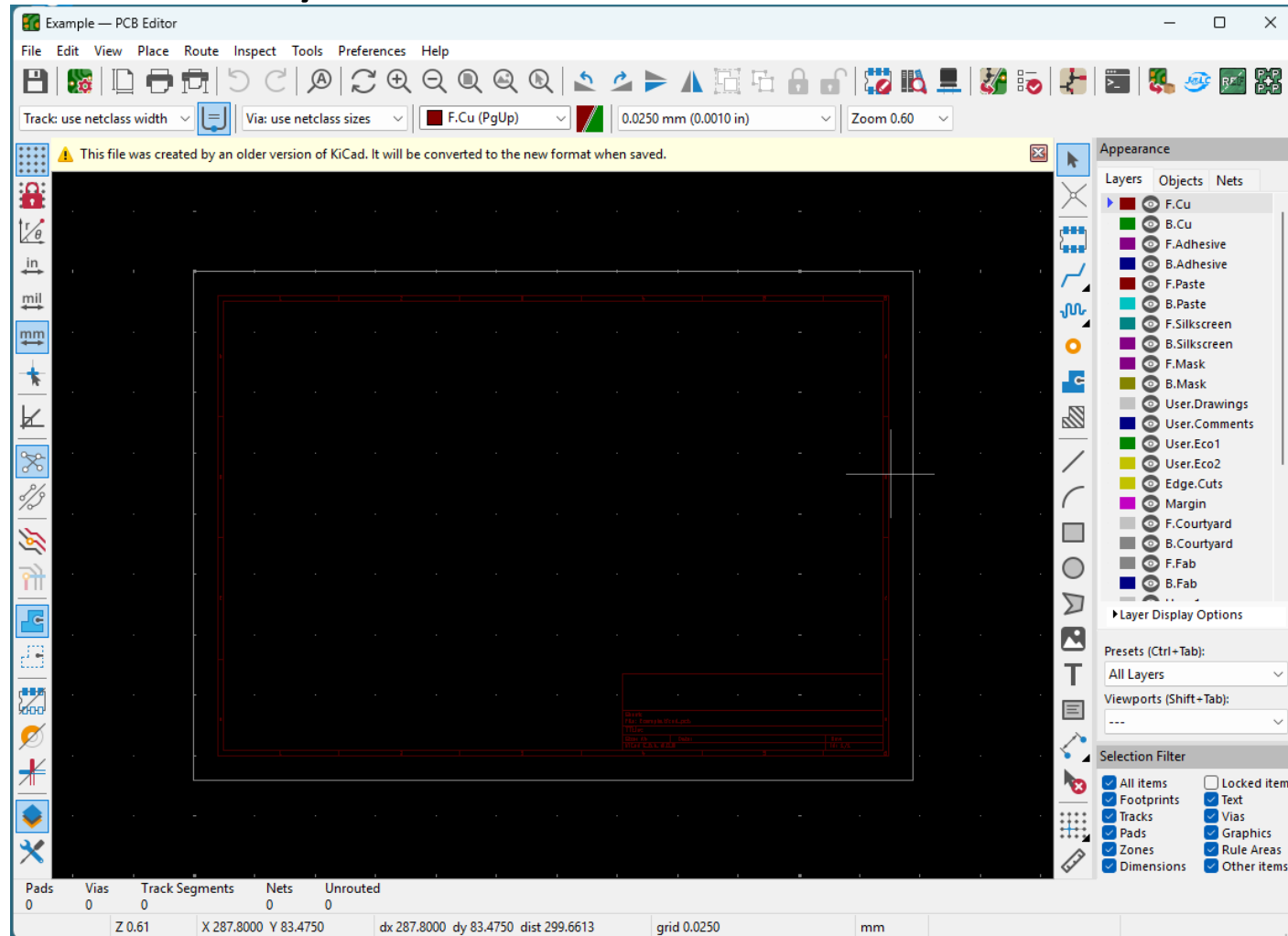
- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

Schematic Layout Editor

Click the
PCB Layout Editor



PCB Layout Editor



- Left – Mode Selection
- Top – Viewing and Misc Tools
- Top – Auxiliary Toolbar
- Right – Drawing Tools
- More Right -- Layers Manager

PCB Layout Editor



- Top Toolbar
 - Identical to the Schematic Layout Editor Toolbar except for a few Icons we aren't using for today's example.

PCB Layout Editor

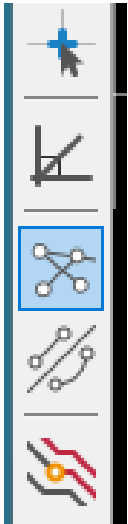
- Left Toolbar – Select Mode



- Turn the Grid On and Off – Blue is On
- Turn on Grid Overrides – Blue is On
- Turn on Polar Coordinates – Blue is On
- Switch Units to Inches – Blue is Selected
- Switch Units to Mils – Blue is Selected
- Switch Units to Millimeters -- Blue is Selected

PCB Layout Editor

- Left Toolbar – Select Mode



- Choose Cursor Shape – Blue is Full Screen
- Limit to Vertical, Horizontal, 45 – Blue is On
- Ratsnest – Blue is On
- Curved Ratsnest Lines – Blue is On
- Toggle Inactive Layers Between Normal and Dimmed

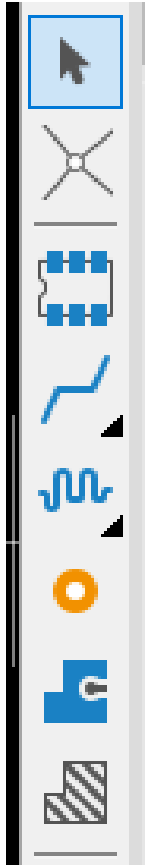
PCB Layout Editor



- Left Toolbar – Select Mode
 - Many more mode settings that we aren't using in today's example.

PCB Layout Editor

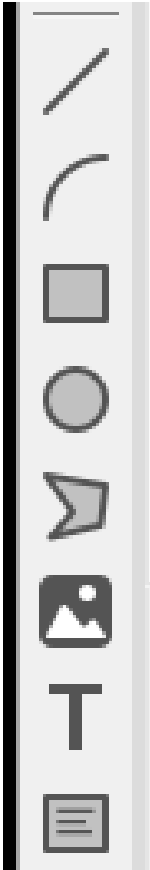
Right Toolbar



- Select Item
- Display Local Ratsnest
- Add Footprints
- Route Tracks
- Tune Length of a Single Net
- Add Vias
- Add Filled Zones
- Add Keepout Areas

PCB Layout Editor

Right Toolbar



- Add Graphic Lines
- Add Graphic Arcs
- Add Graphic Rectangle
- Add Graphic Circle
- Add Graphic Polygon
- Add Bitmap Image
- Add Text
- Add Text Box

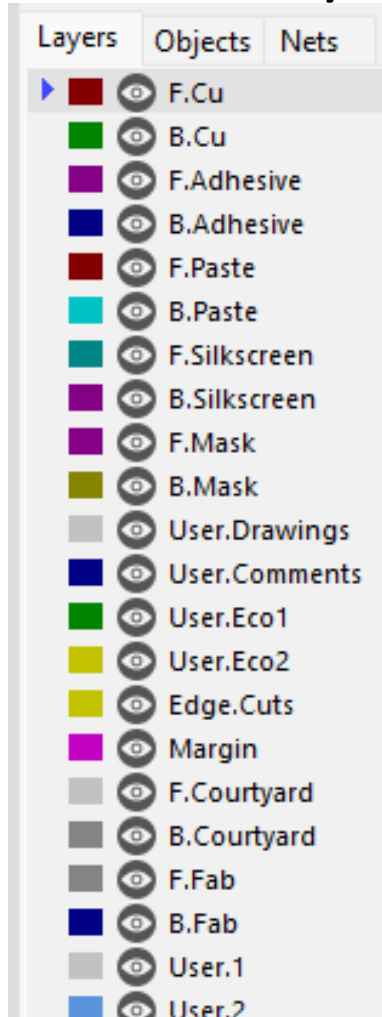
PCB Layout Editor

Right Toolbar



- Add an Aligned Linear Dimension
- Deleted Selected Item
- Place Grid Origin Point
- **Measure Distance**

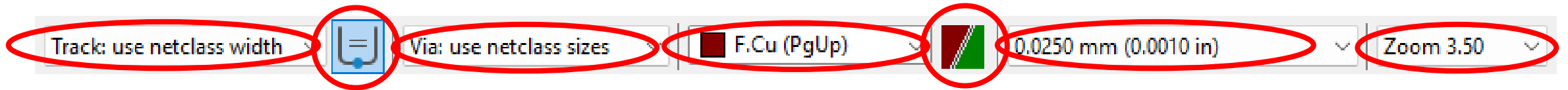
PCB Layout Editor



Layer Manager

- Controls Display of each Layer in the Drawing Area
- The Blue Arrow Defines Where Operations Occur
 - Also shown on Auxiliary Toolbar

PCB Layout Editor



Auxiliary Toolbar Mode Selections

- Track Width
- Use Existing Track Width – Blue is On
- Via Size
- Layer to Route Traces On
- Pair of Layers to Route Traces On
- Grid Step Size
- Zoom Step Size

KiCad – Work Flow

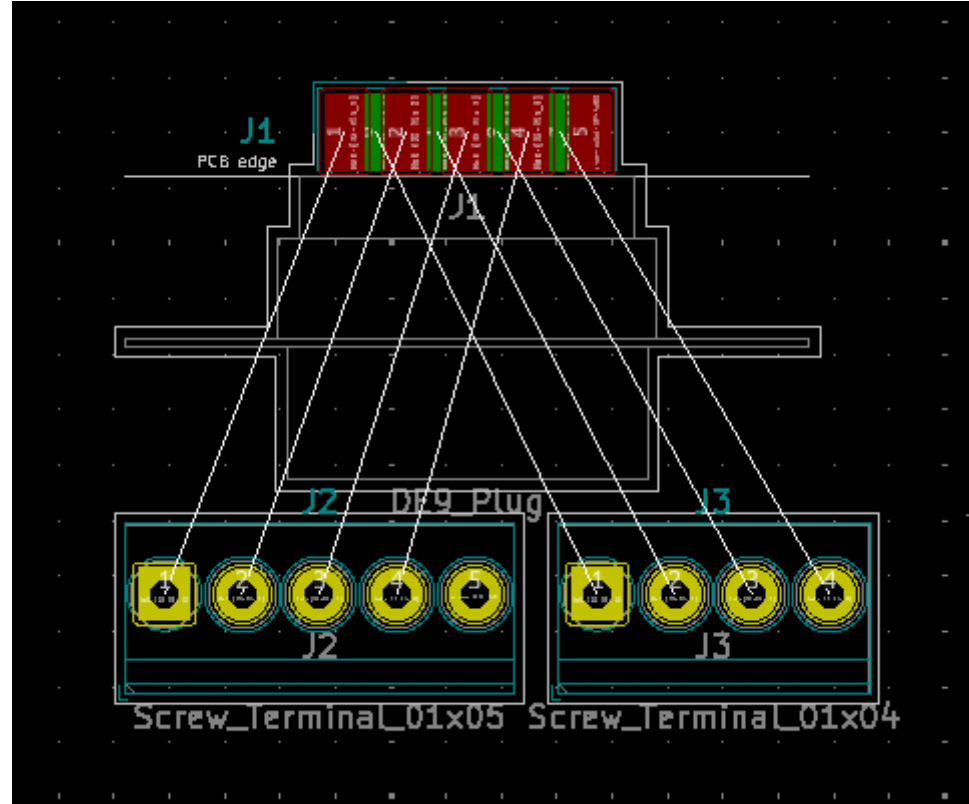
- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

PCB Layout Editor

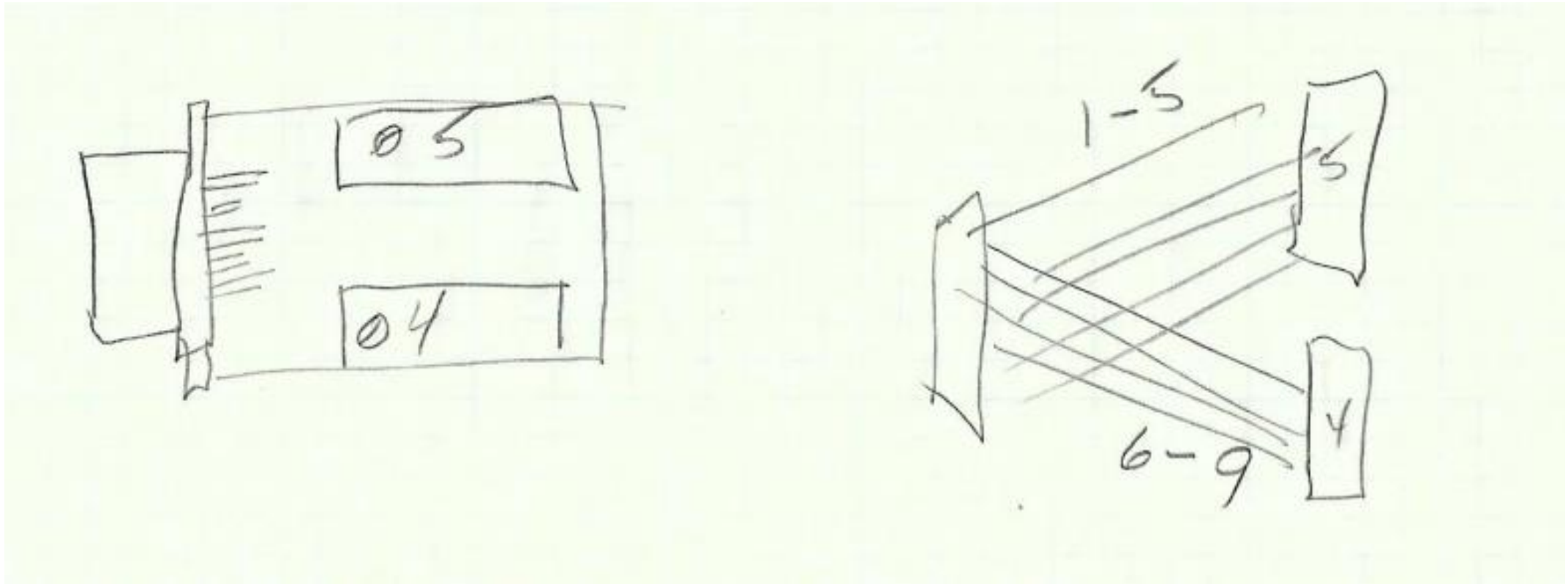
Click to Update PCB
from Schematic



Footprints and
Ratsnest From
Schematic are
Displayed on PCB

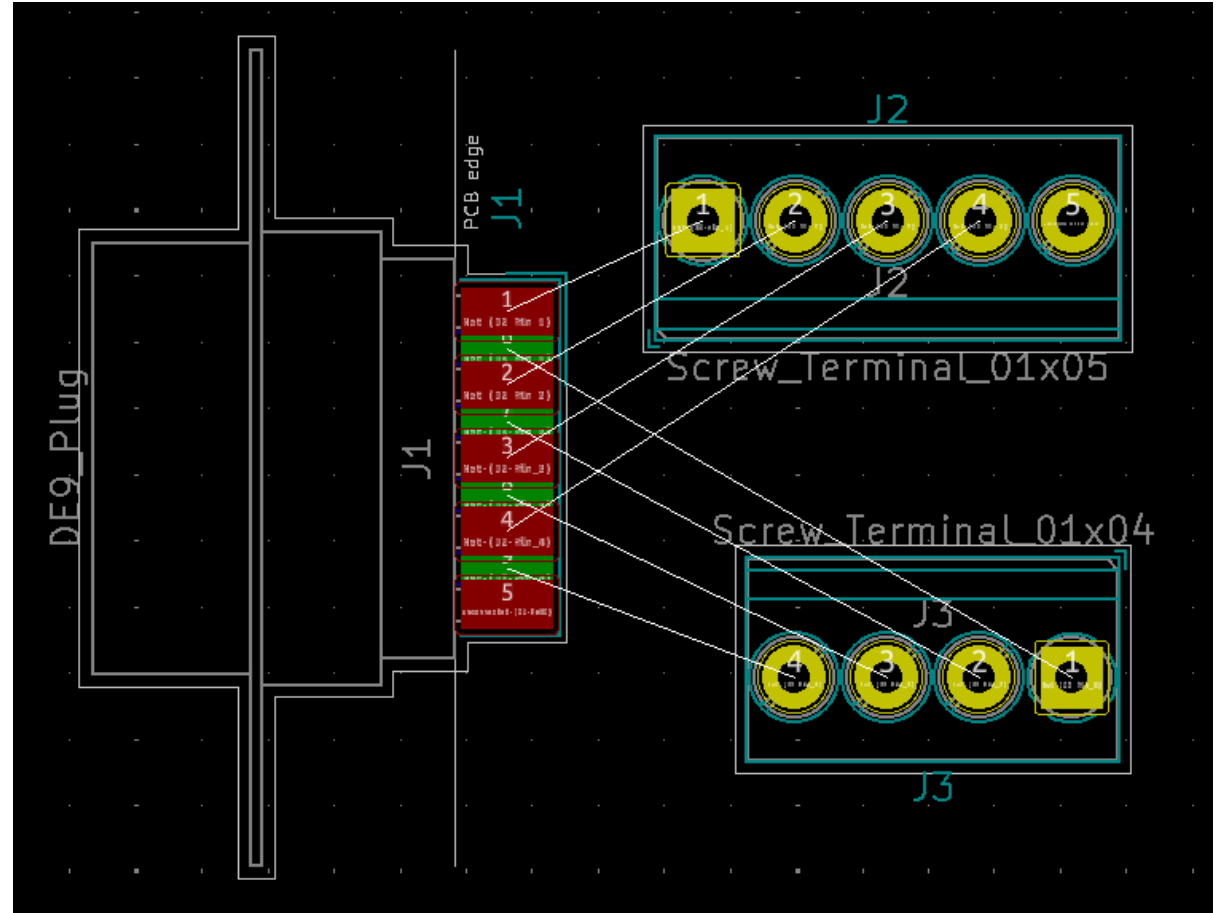


Example Project – Hand Sketch



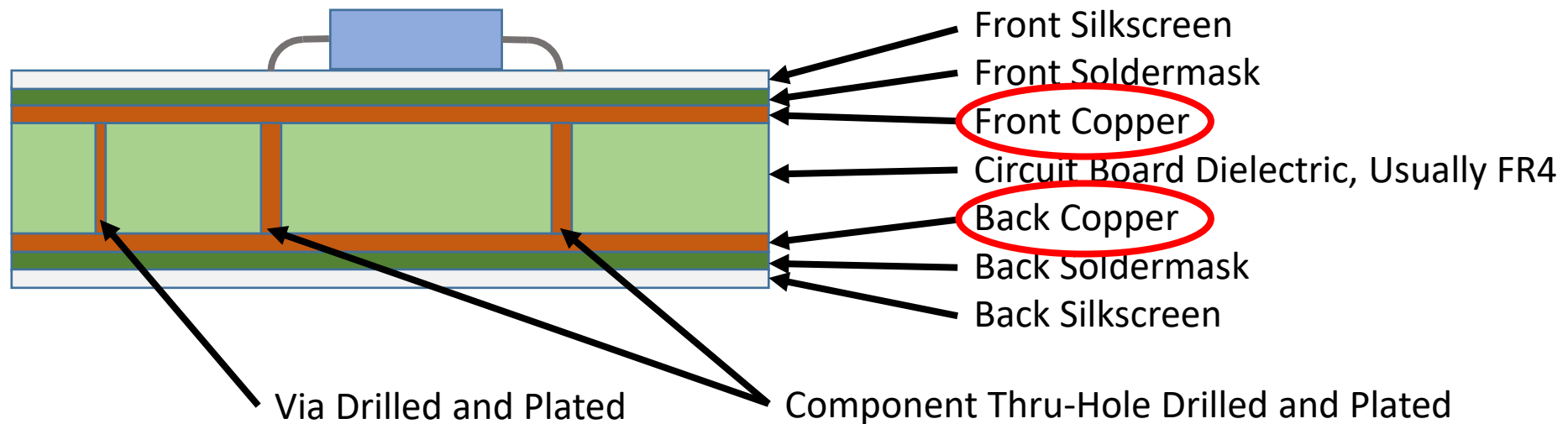
PCB Layout Editor

- Select and Move each Component to Position
- Press R to Rotate
- Try to “Untwist” the Rats to Make Routing Easier
- Note:
 - Red Pads are on the Front
 - Green Pads are on the Back

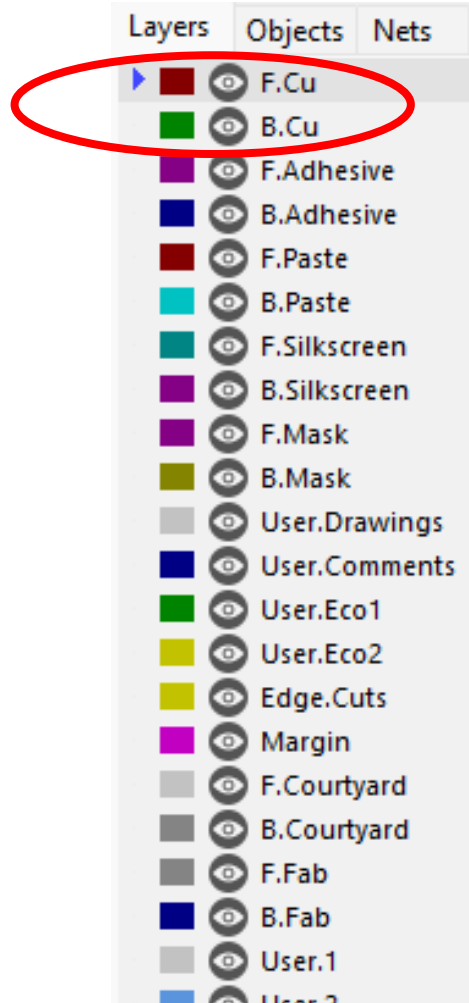


Anatomy of a PCB

- Many Steps to Manufacturing a PCB
- “Gerber” Files Used in the Manufacturing Process
- KiCad Creates Gerber Files
- “Layers” Refers to the Number of Copper Layers, Always Even, Up to 30



PCB Layout Editor

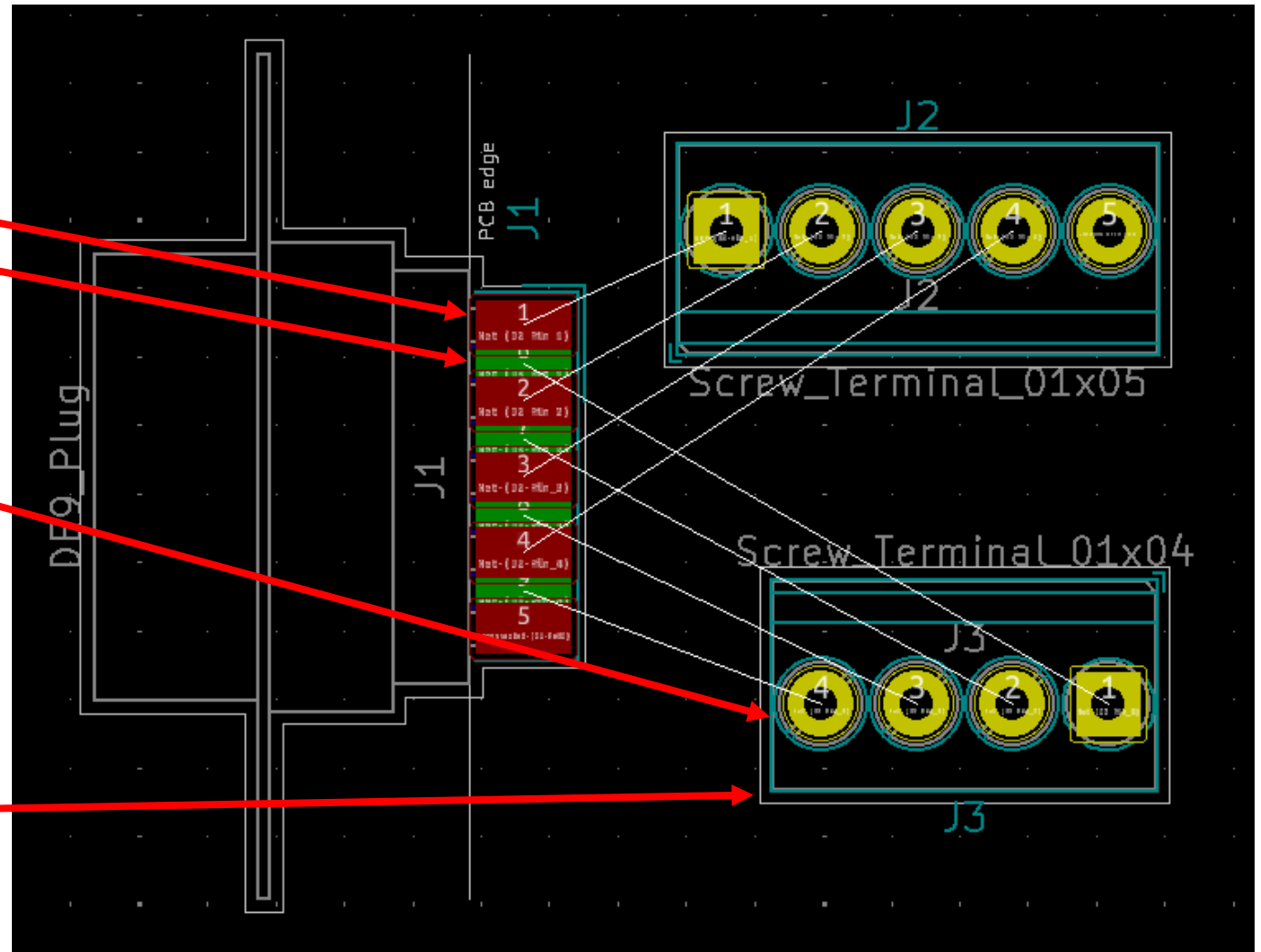


Layer Manager

- Control Display of each Layer in the Drawing Area
- The Blue Arrow Defines Where Modifications Occur

PCB Layout Editor

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	



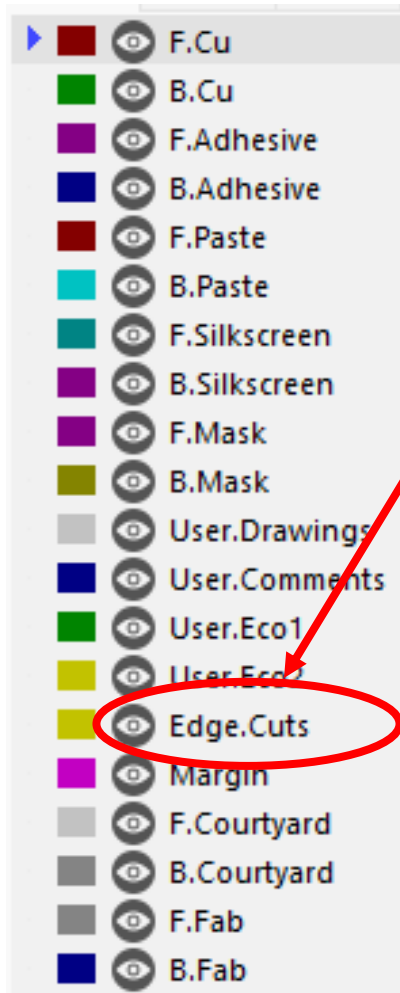
KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

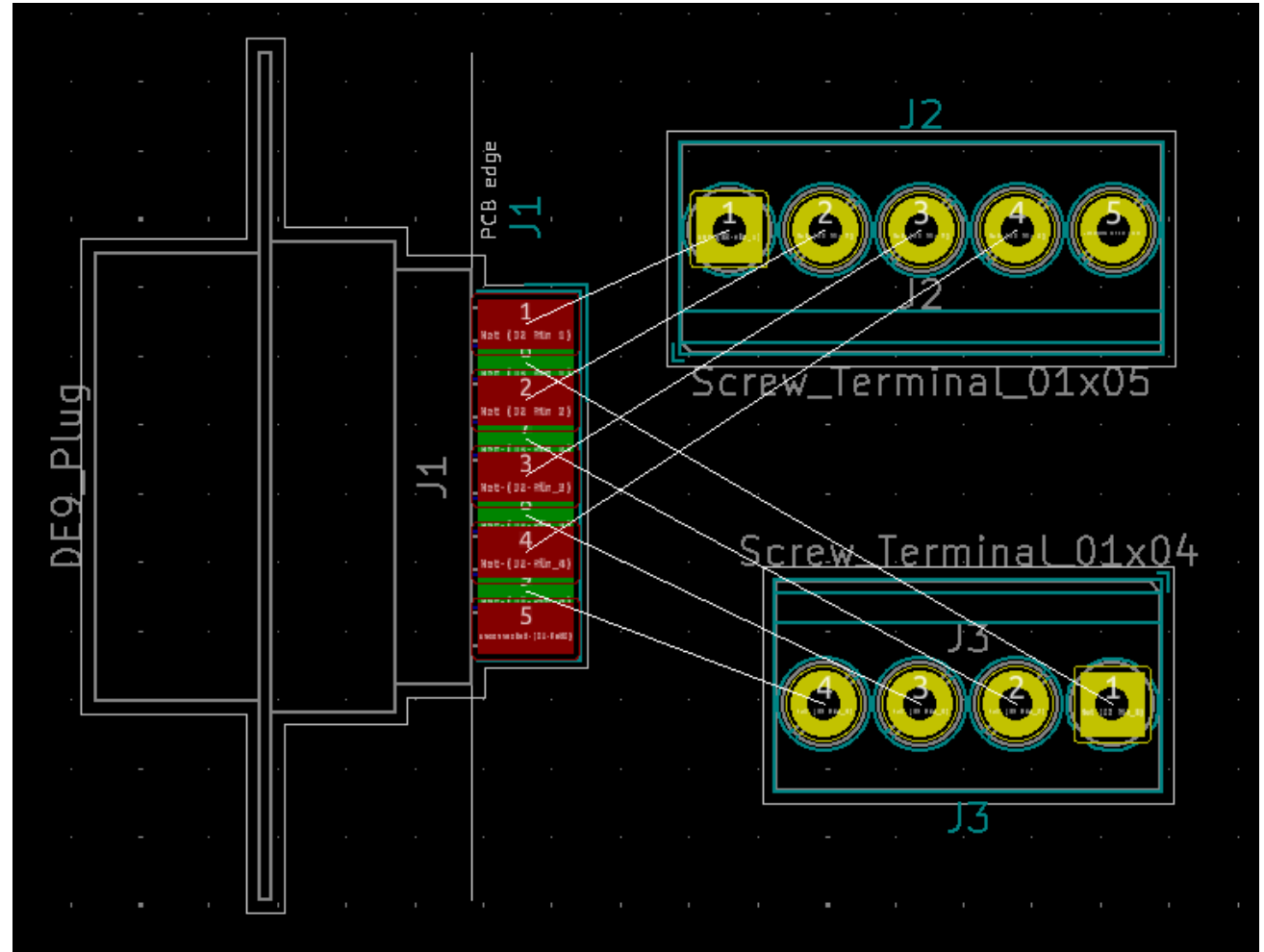
KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

PCB Layout Editor

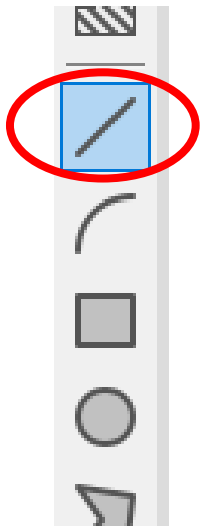


Select the
Edge.Cuts Layer

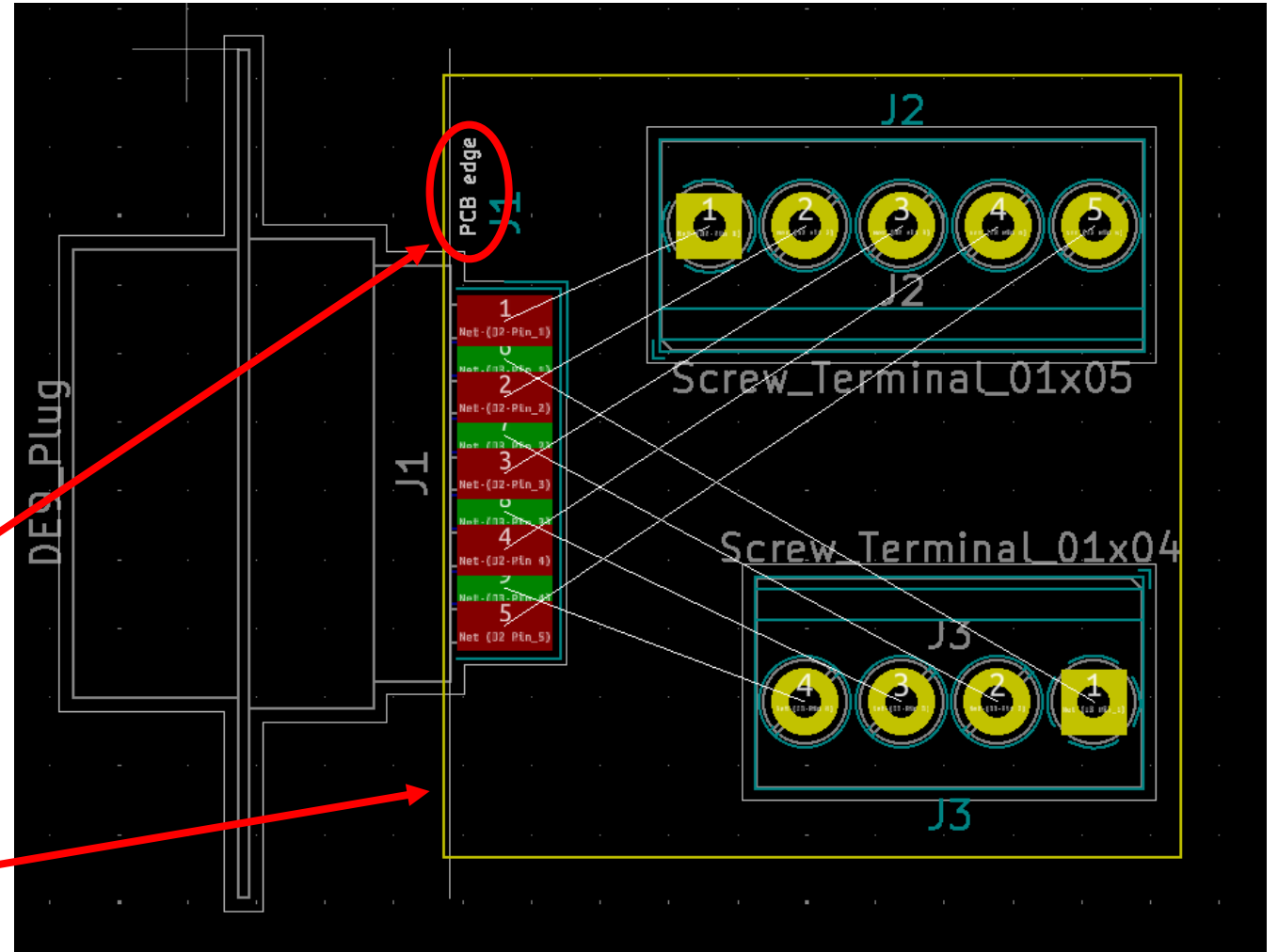


PCB Layout Editor

- Click Add Graphics Lines

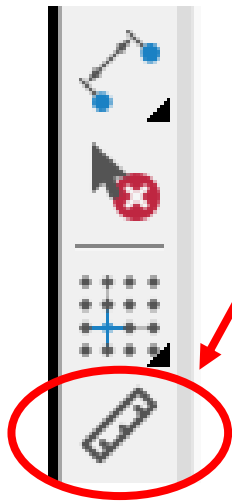


- Note “PCB Edge”
- Draw a Complete Box Around Components

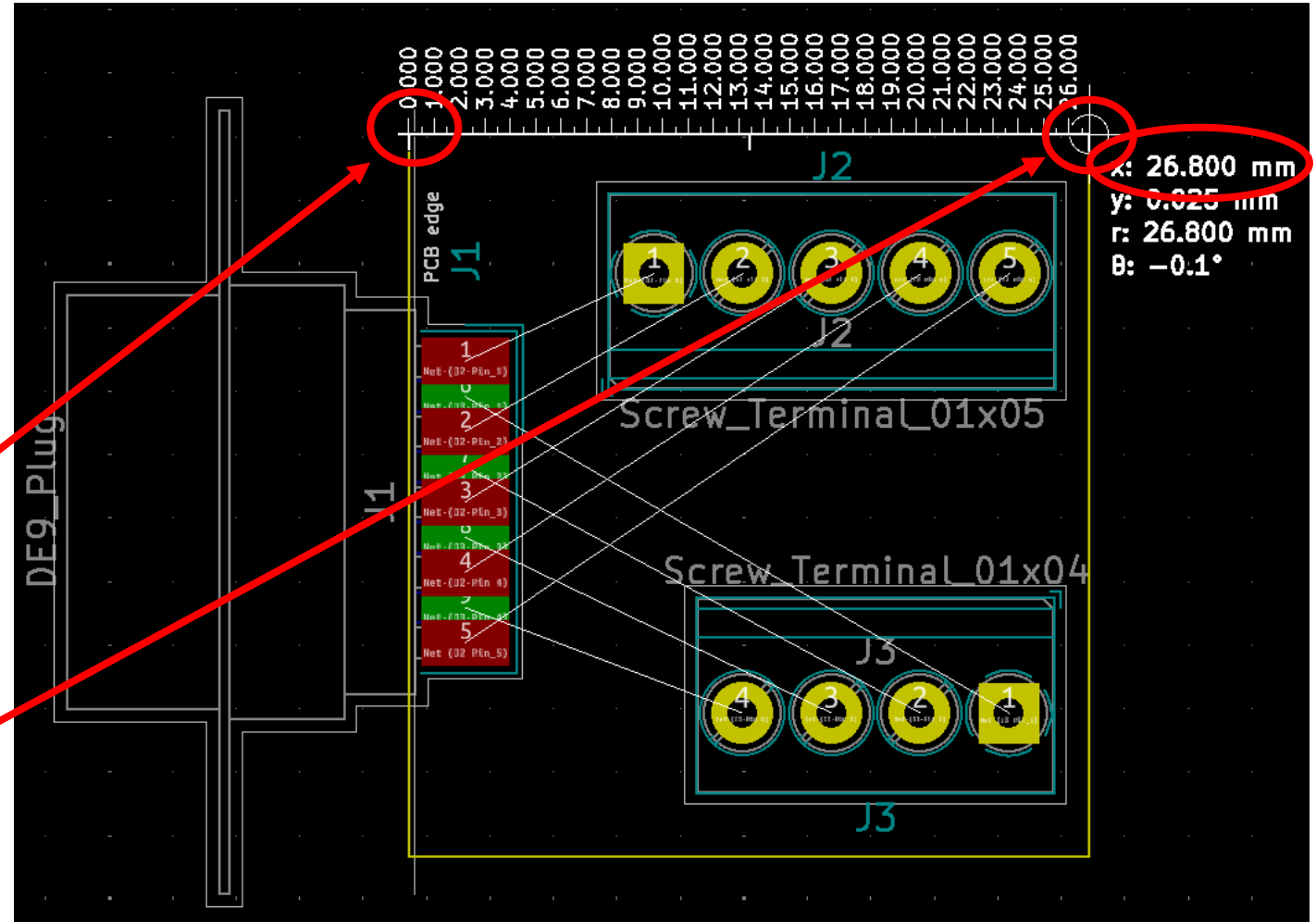


PCB Layout Editor

Measure Board Width.
Use the Measure Tool.

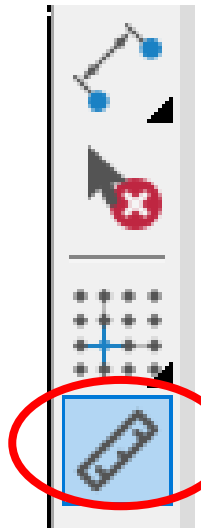


Click the Start and End
of the Measurement.

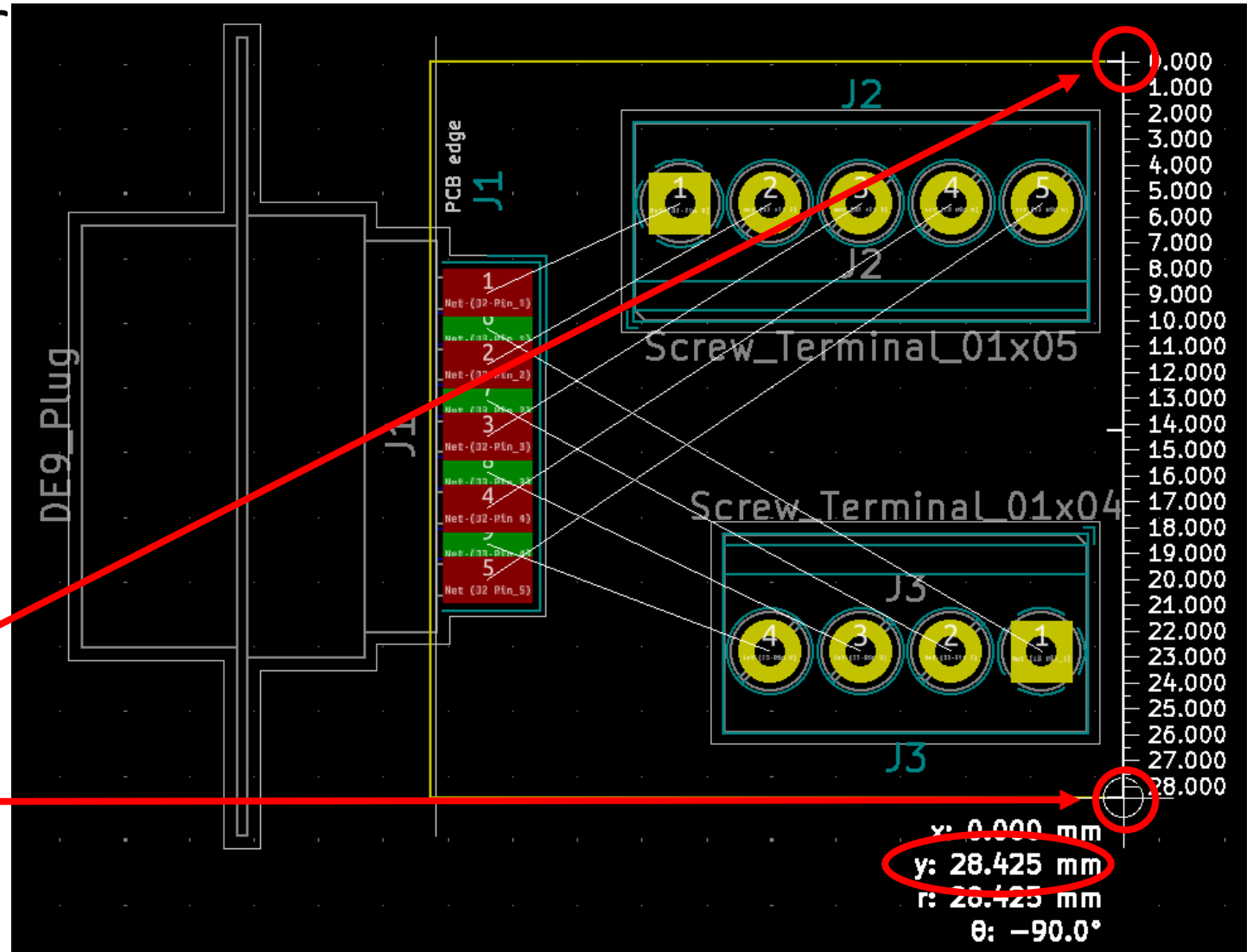


PCB Layout Editor

Measure Length. Use the Measure Tool.



Click the Start and End of the Measurement.



KiCad – Work Flow

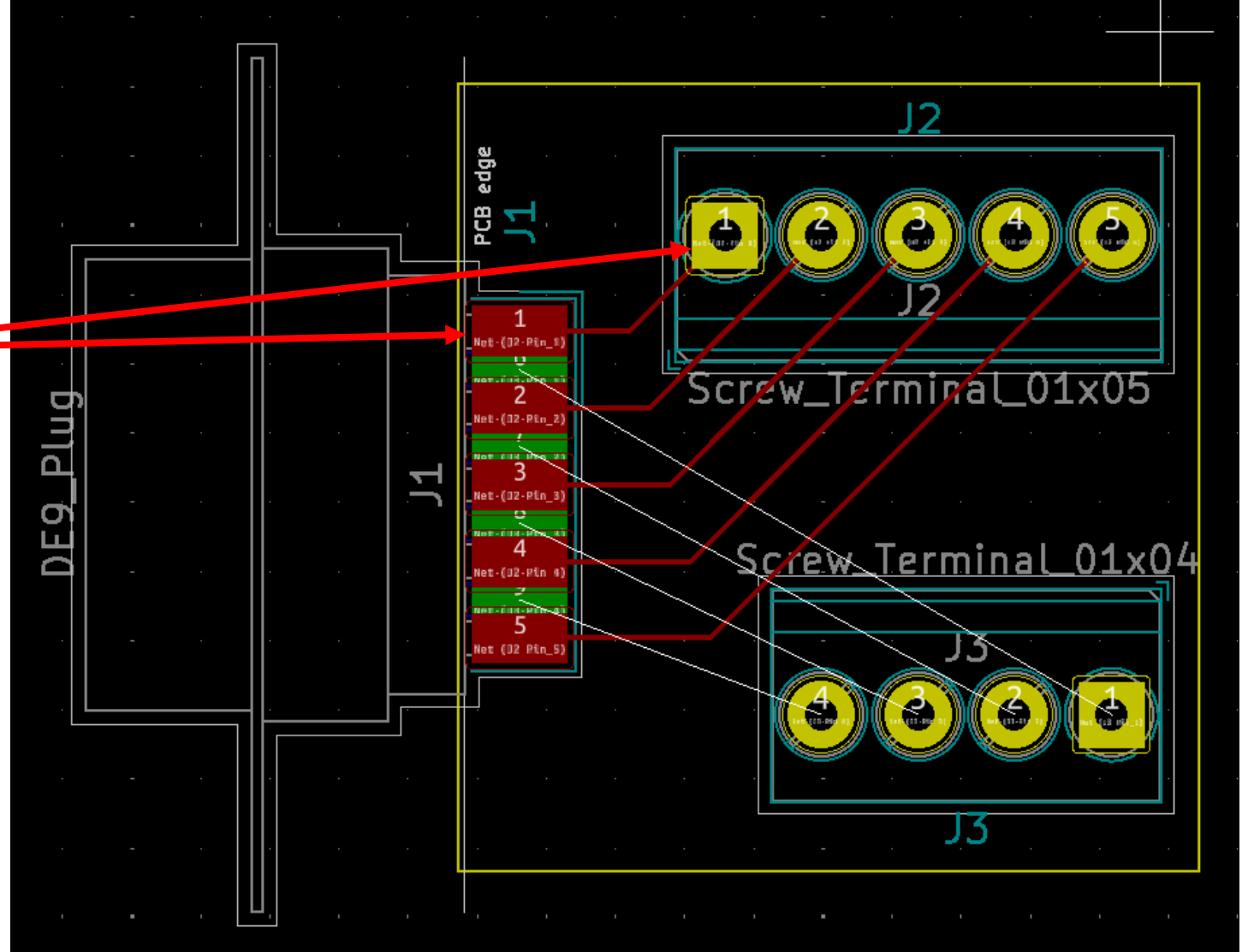
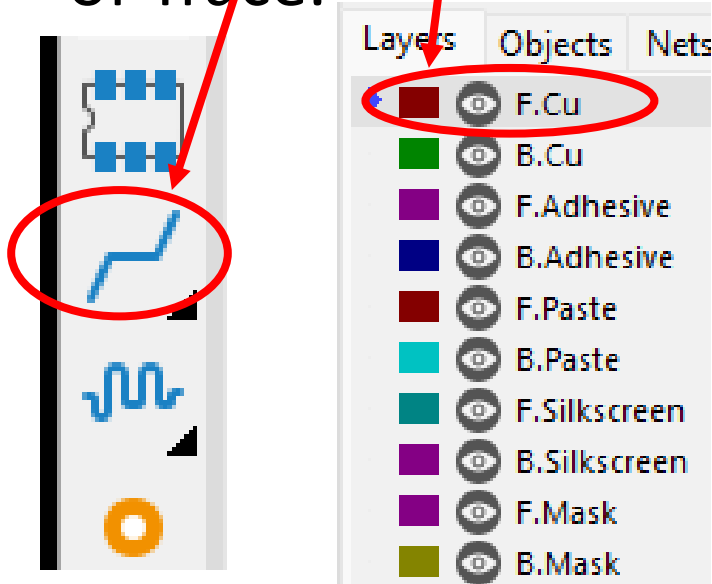
- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
 - Generate Netlist
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

PCB Layout Editor

Click Route Track

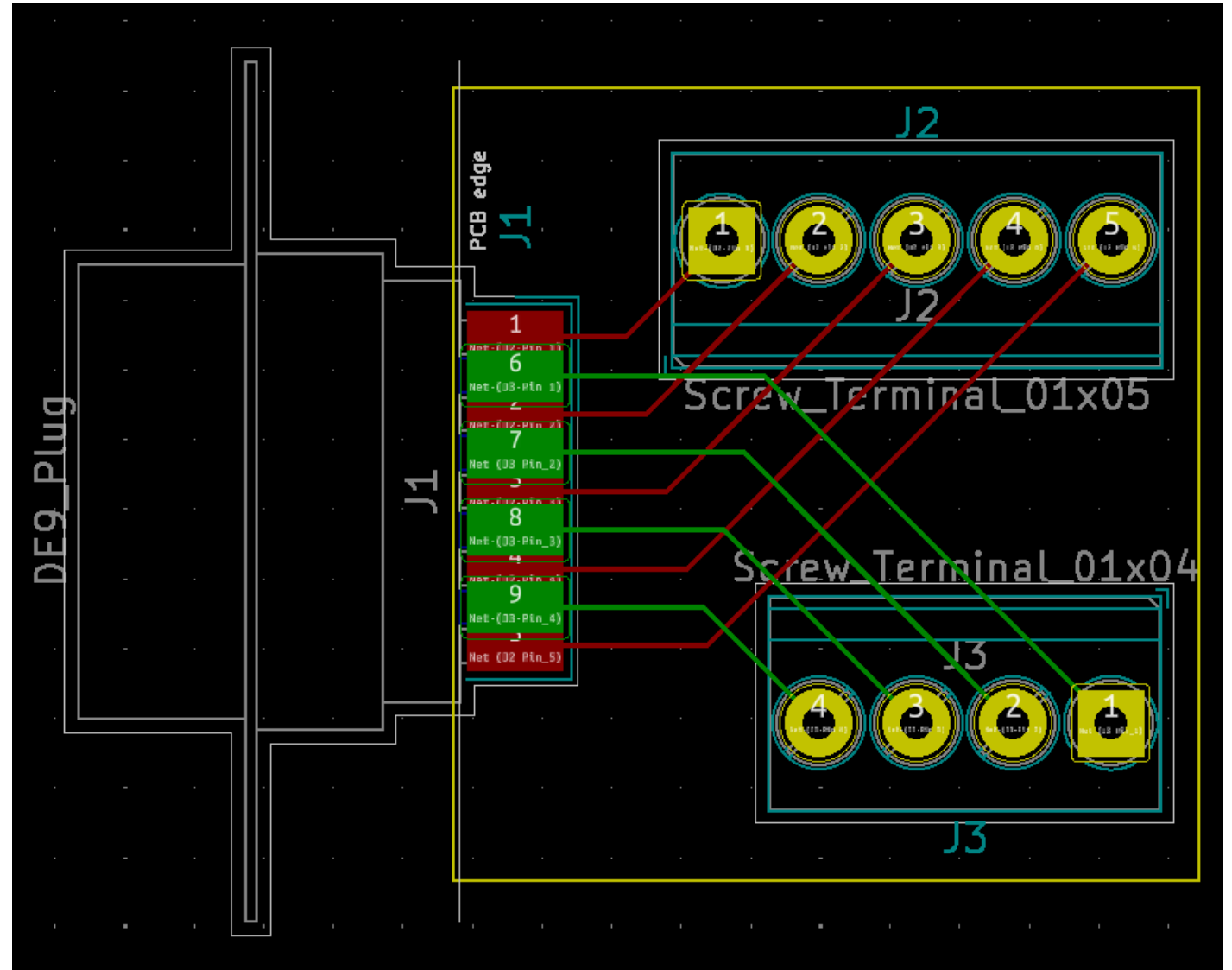
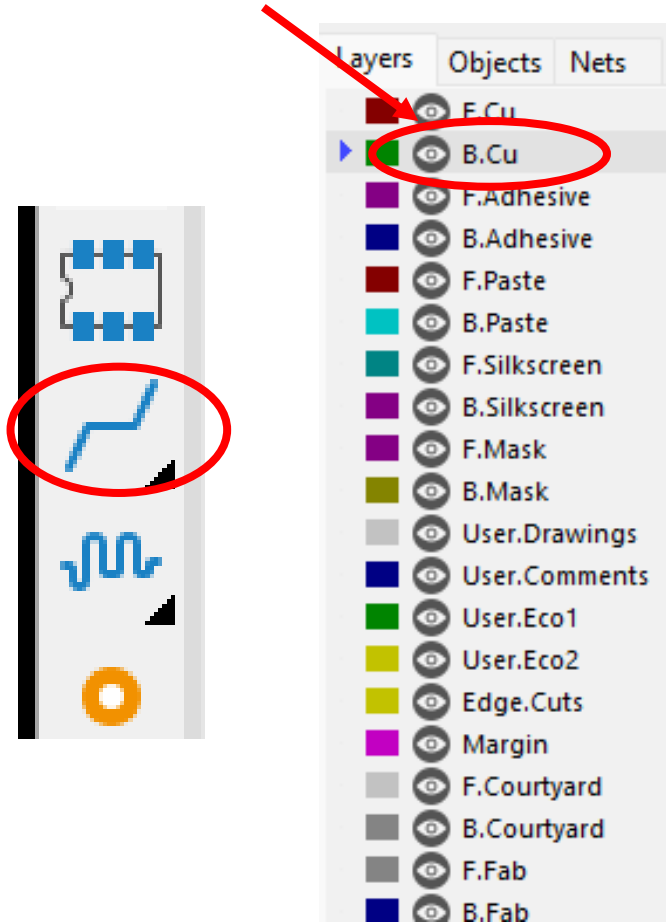
Click Front Copper

Click Start and End of Trace.



PCB Layout Editor

Now the Back

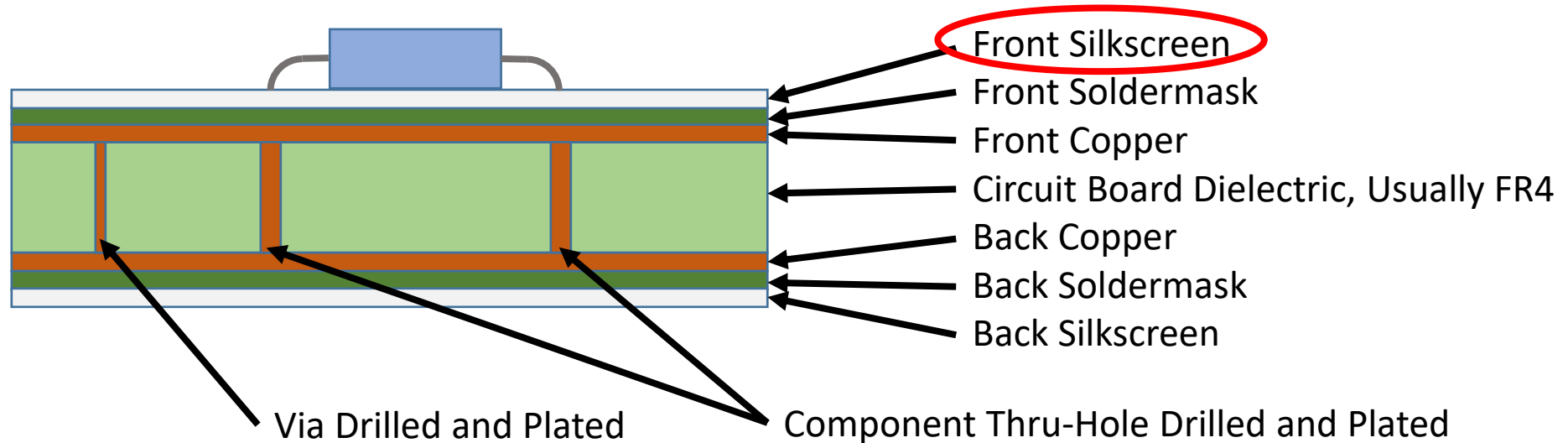


KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
 - Generate Netlist
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

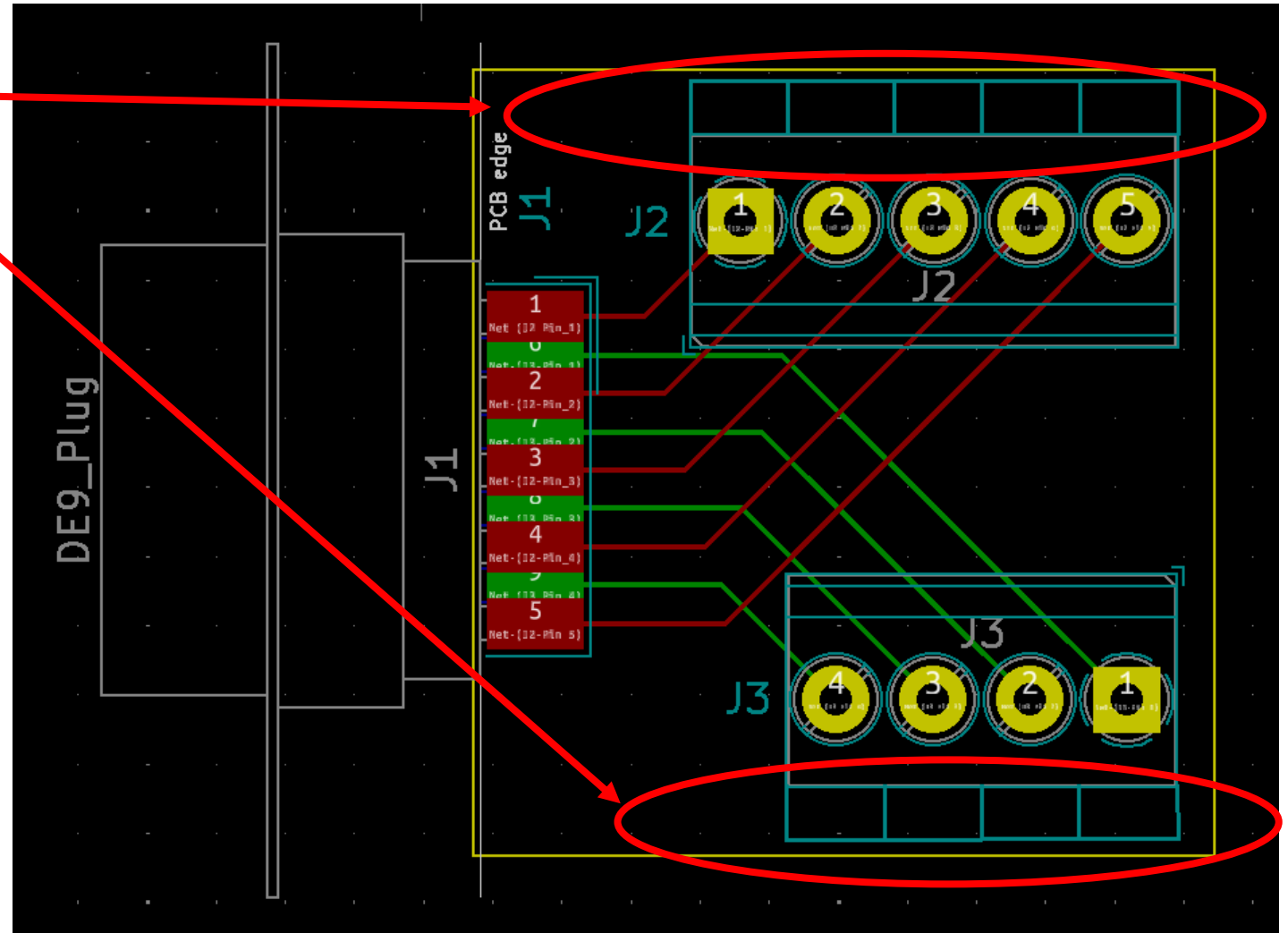
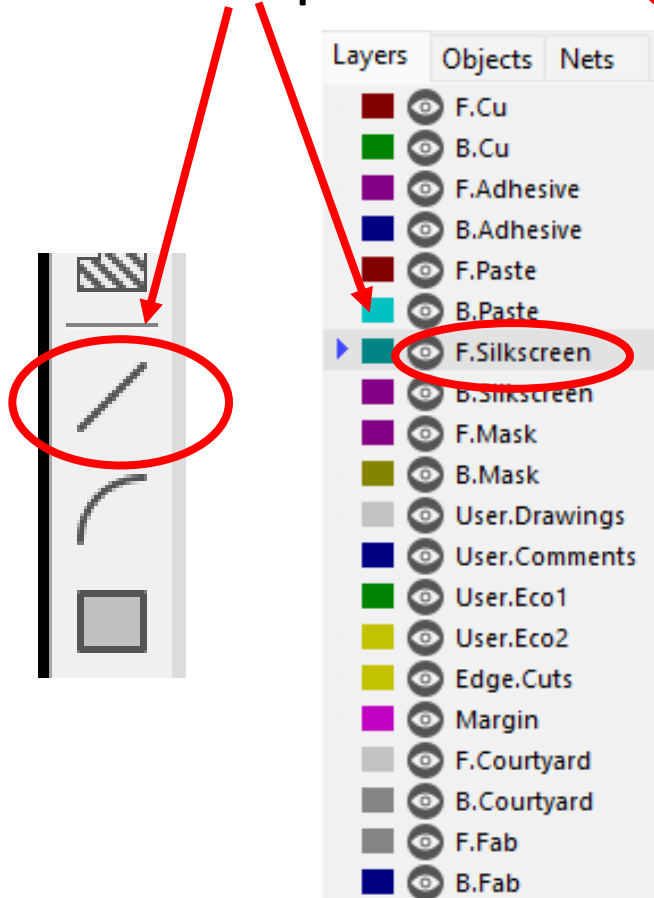
Anatomy of a PCB

- Many Steps to Manufacturing a PCB
- “Gerber” Files Used in the Manufacturing Process
- KiCad Creates Gerber Files
- “Layers” Refers to the Number of Copper Layers, Always Even, Up to 30



PCB Layout Editor

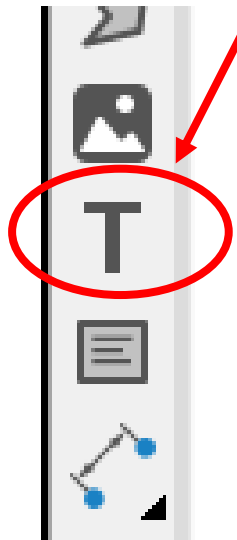
Add Graphics Line



PCB Layout Editor

Click Add Text Icon

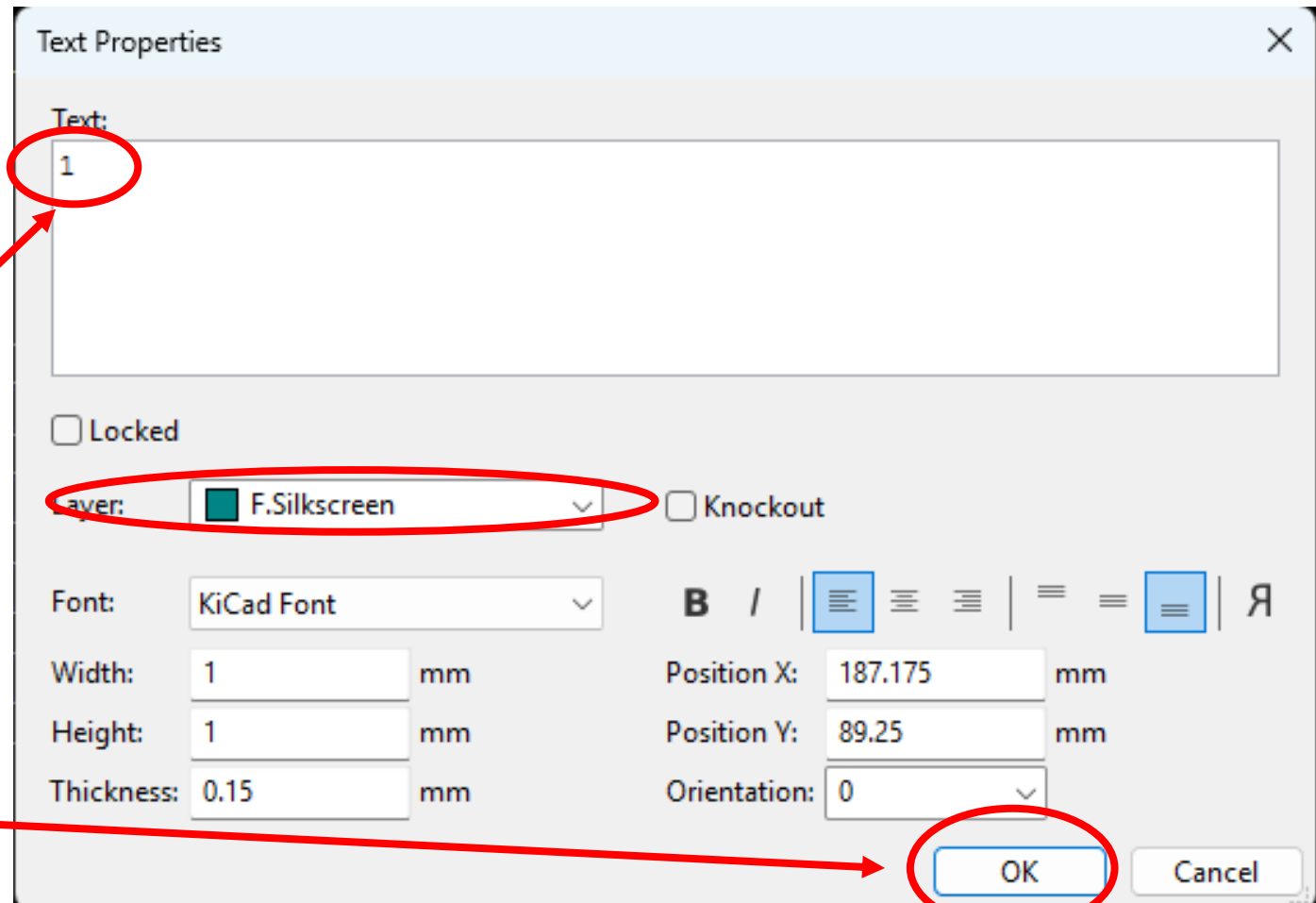
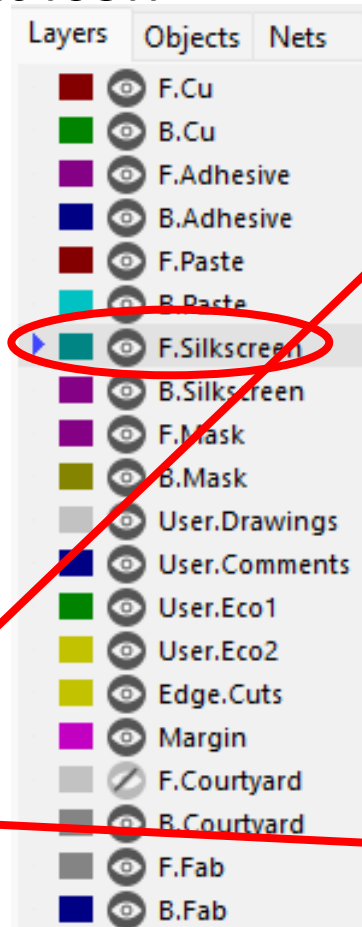
Click Sheet



Add Text

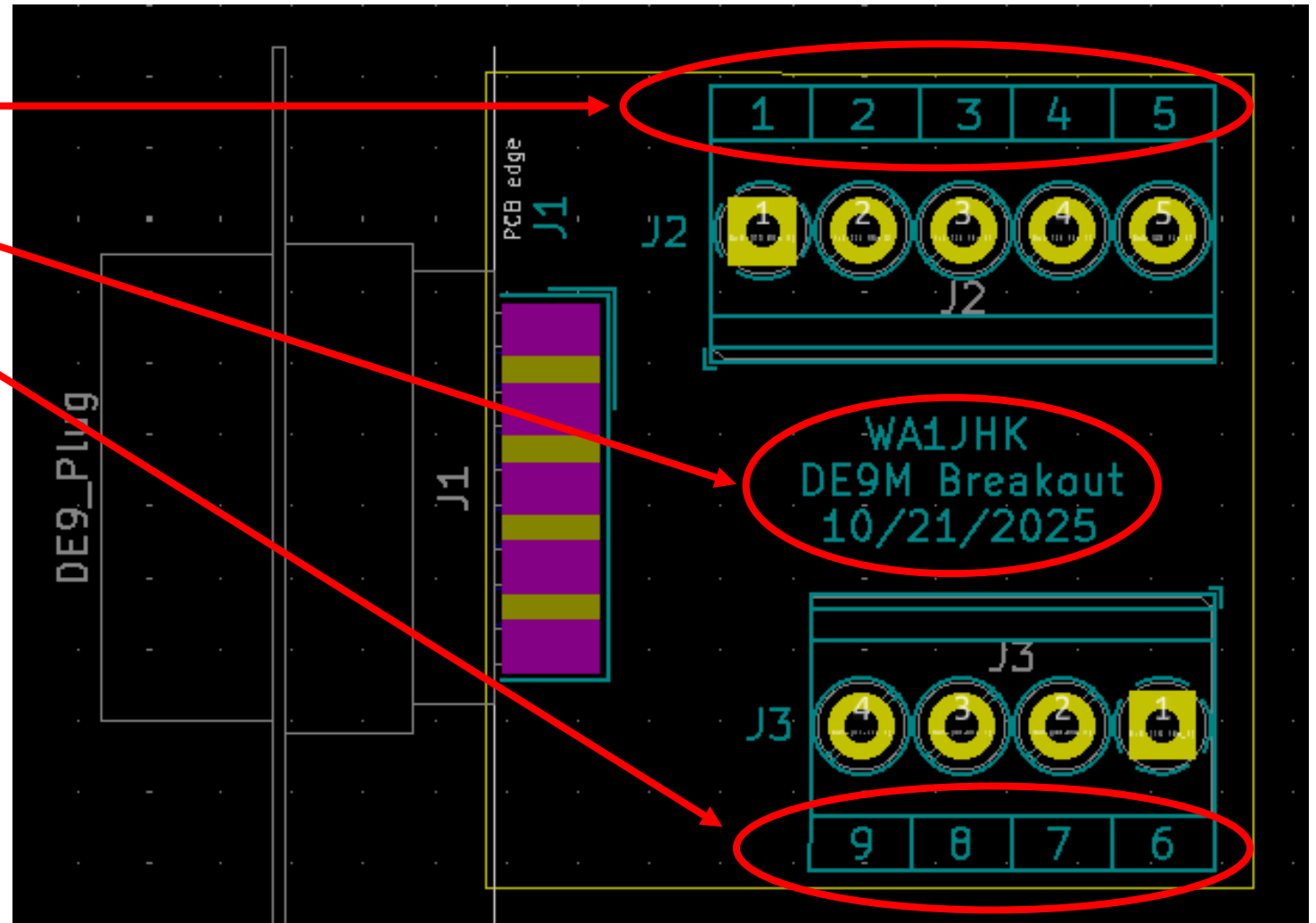
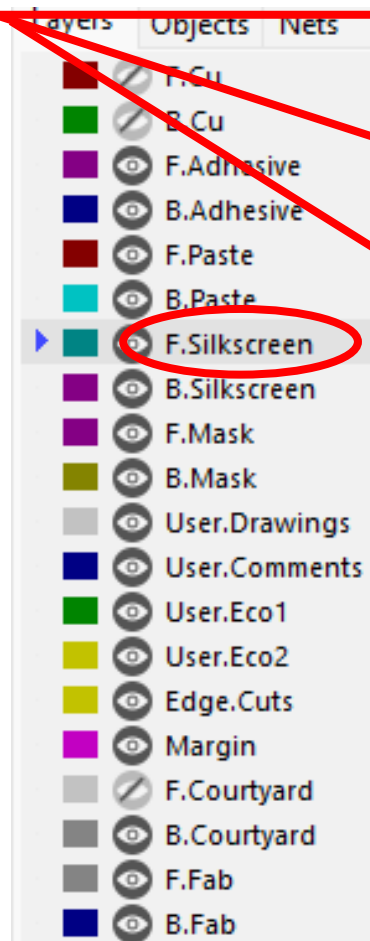
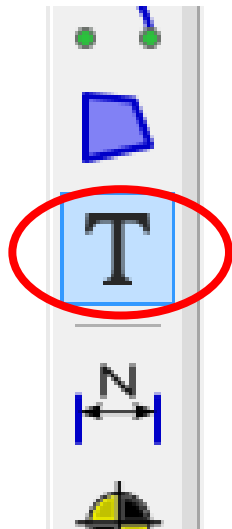
Click OK

Click Sheet



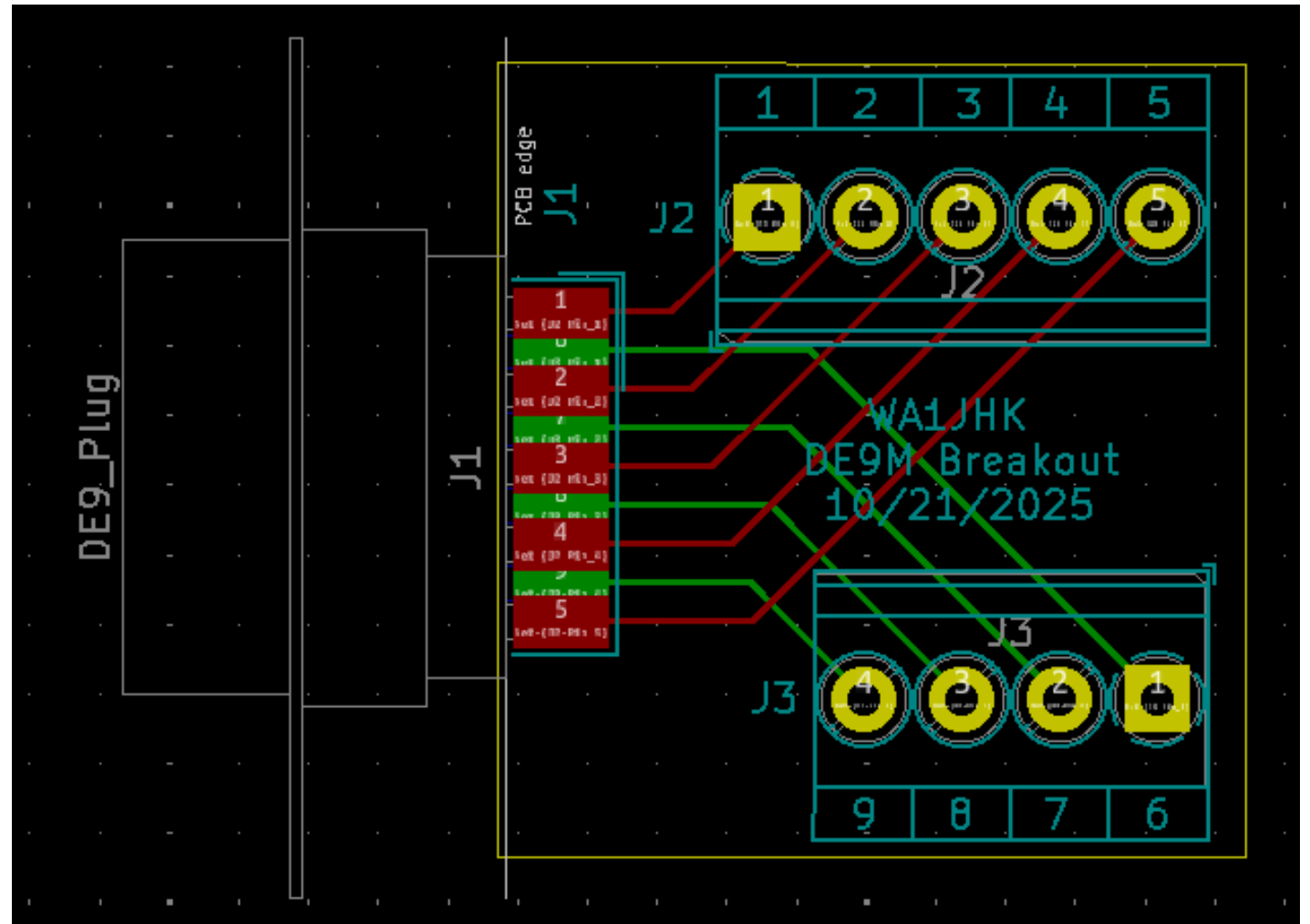
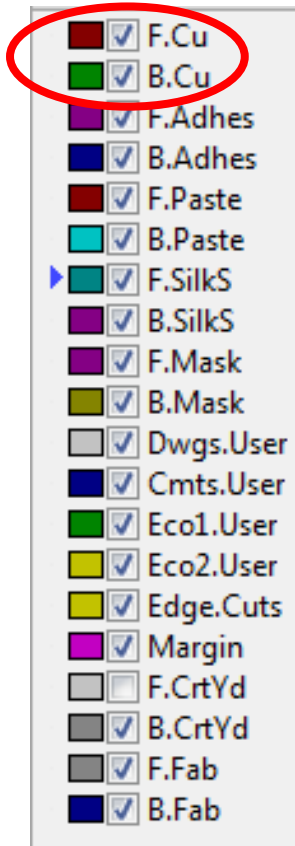
PCB Layout Editor

Add Text



PCB Layout Editor

Copper Layers Back On



KiCad – Work Flow

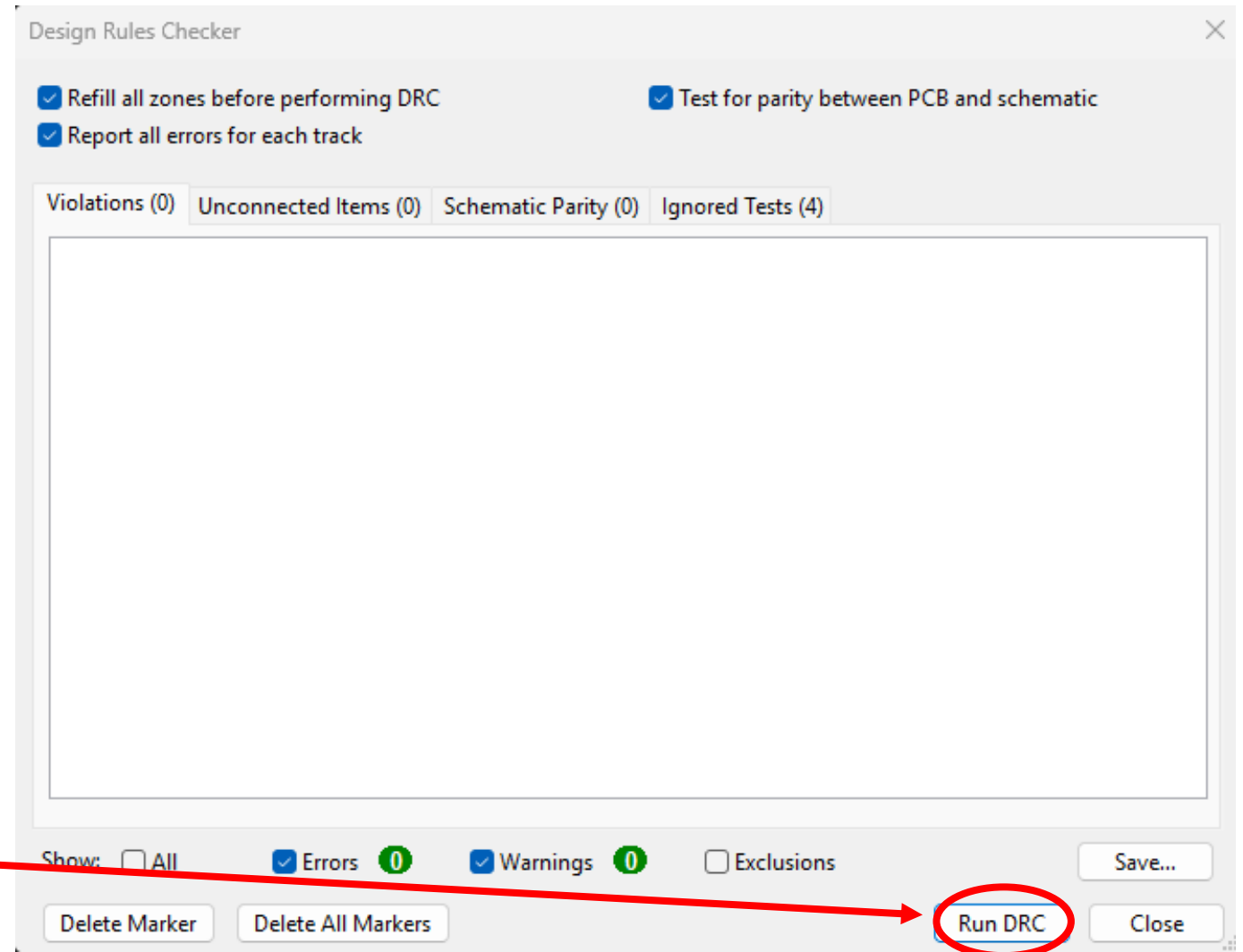
- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
 - Generate Netlist
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

PCB Layout Editor

Click Design Rule Check



Click Run

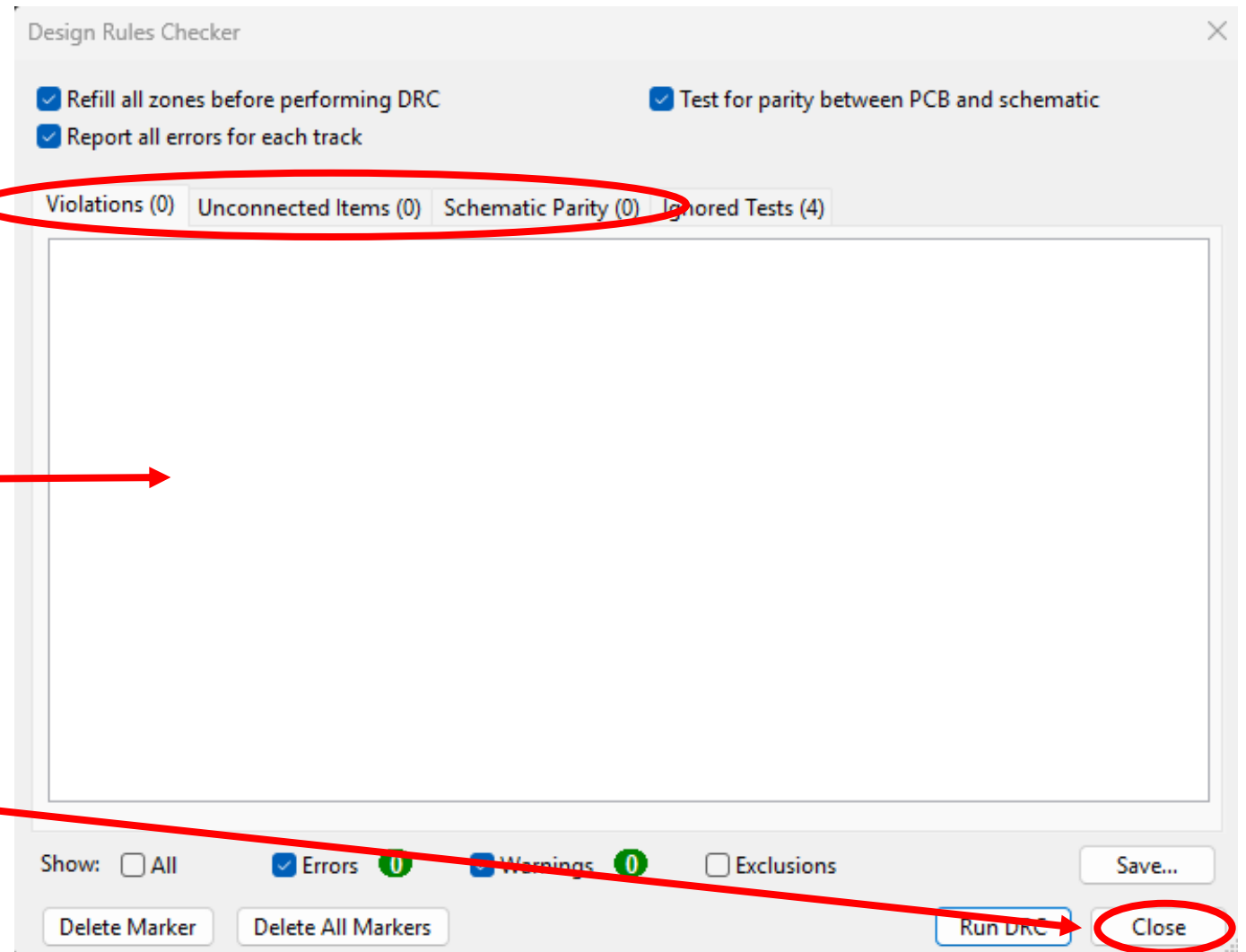


PCB Layout Editor

Check Errors

Check Error Log

Click Close



KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
 - Generate Netlist
- PCB Layout Editor
 - Update PC from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

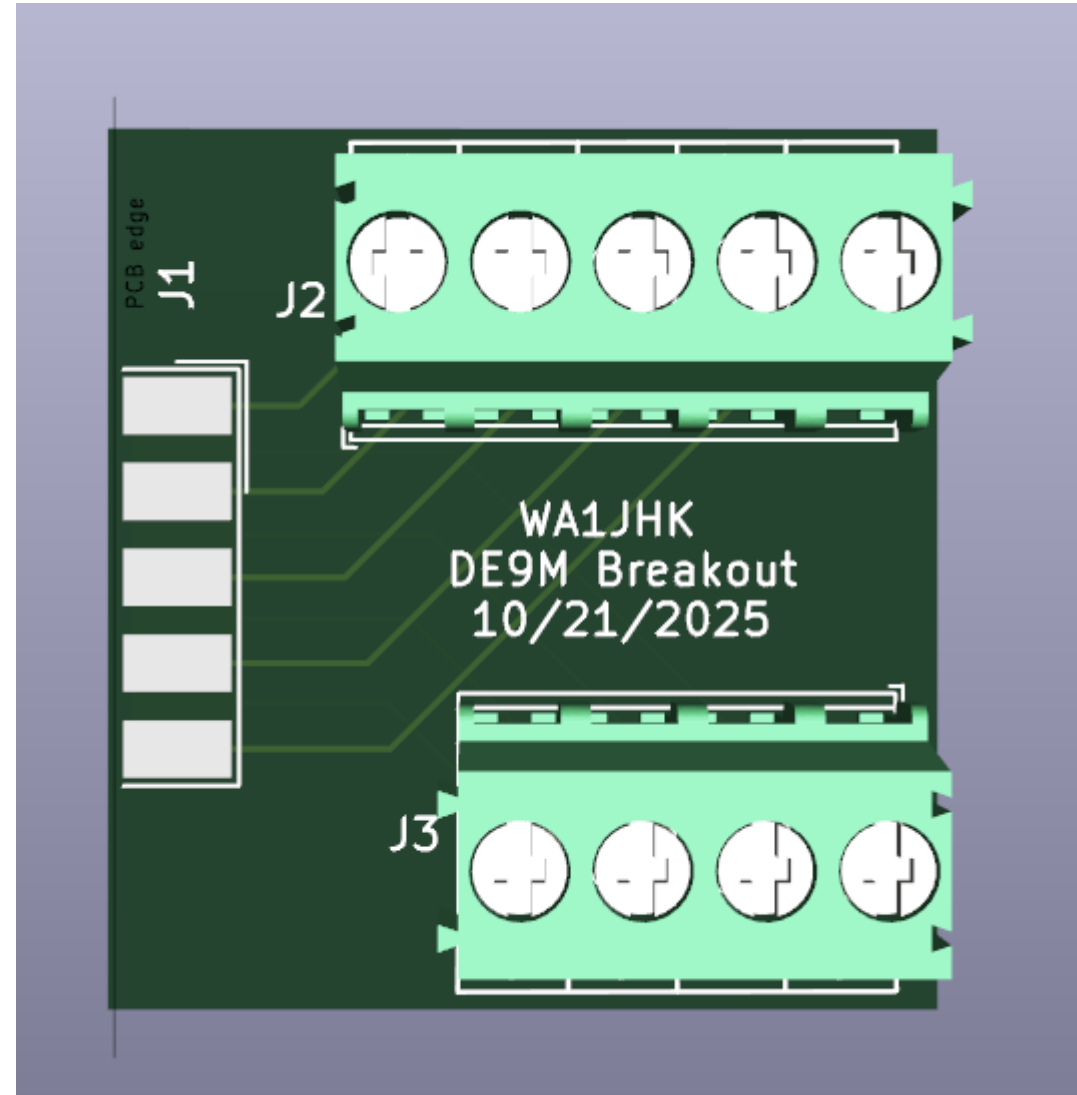
PCB Layout Editor

Press ALT-3

3D Viewer Displays in
a new window.

You can resize and
spin to any view.

Use the Toolbar or
click and drag.



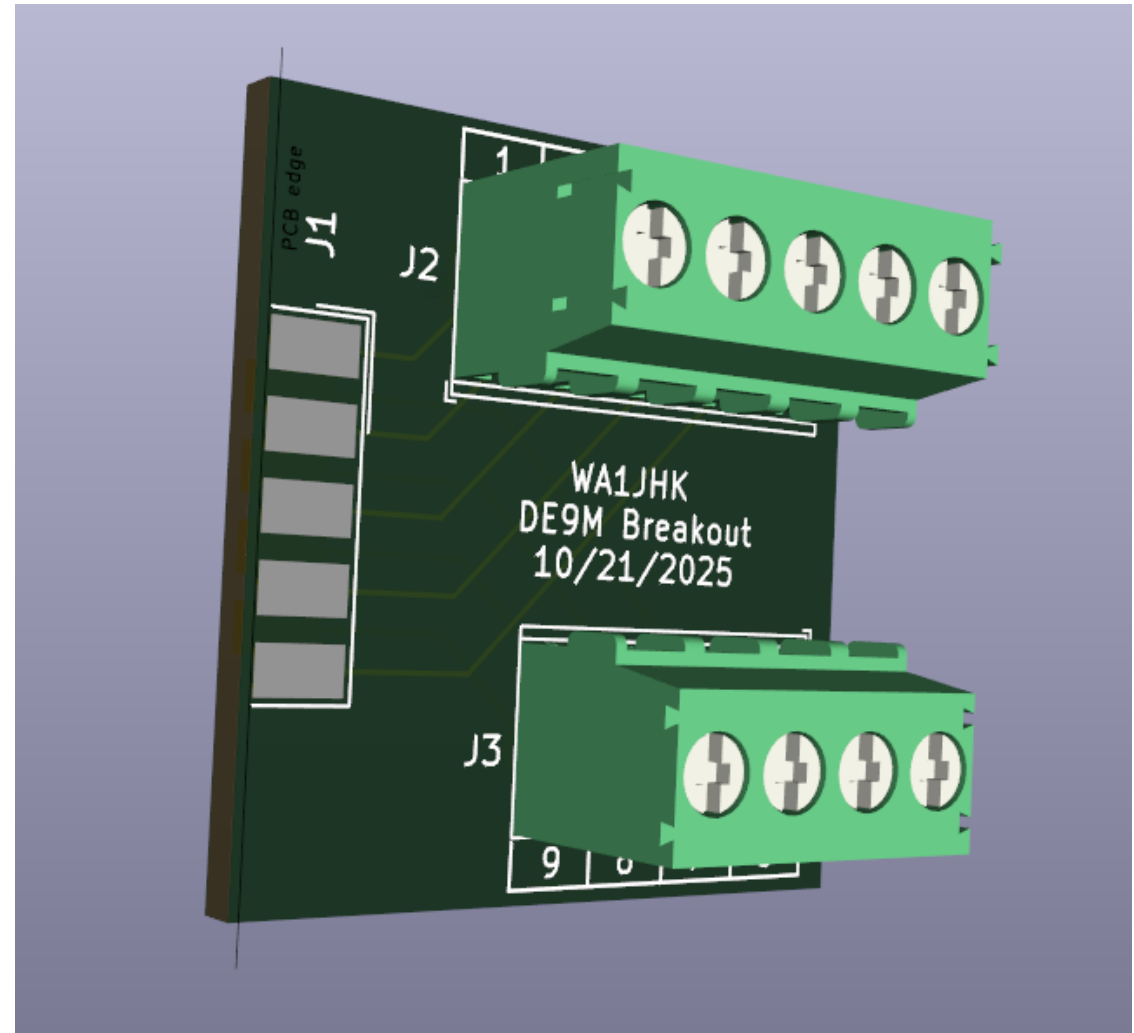
PCB Layout Editor

Press ALT-3

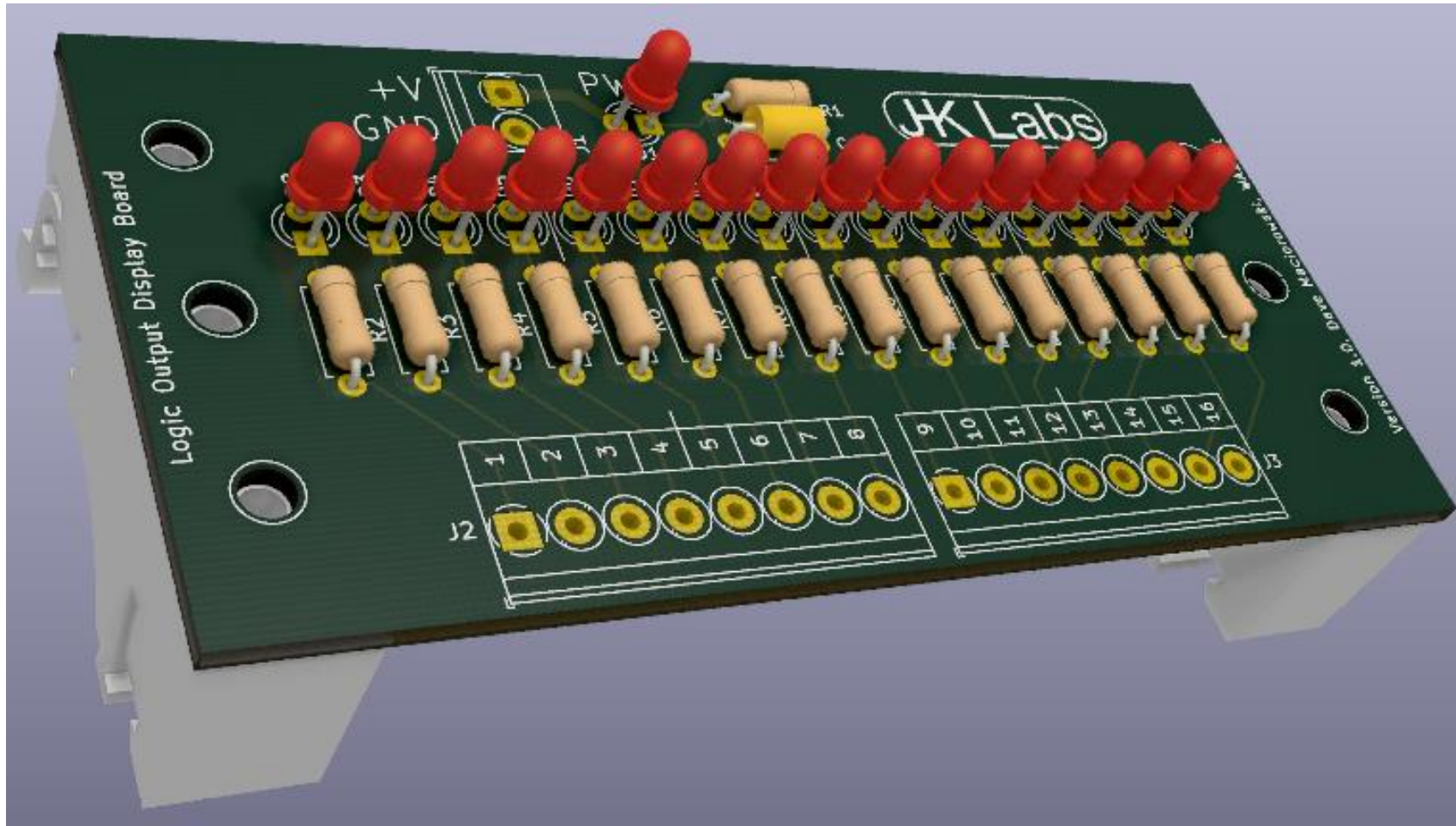
3D Viewer Displays in
a new window.

You can resize and
spin to any view.

Use the Toolbar or
click and drag.



PCB Layout Editor



KiCad – Work Flow

- Hand-Sketched Schematic
 - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
 - Set Page Size
 - Place Symbols from Library
 - Optional Symbol Editor
 - Annotate Schematics
 - Electrical Rule Check
 - Associate Footprint With Component
 - Generate Netlist
- PCB Layout Editor
 - Update PCB from Schematic
 - Mounting Holes, Mechanical
 - Symbol and Footprint Library
 - Optional Footprint Editor
 - Draw Edge Cuts
 - Route the Board
 - Add Silkscreen
 - Design Rule Check
 - 3D Viewer
 - Plot Gerbers
- Gerber Viewer
 - Layer Reviews
- Upload to JLC PCB

PCB Layout Editor

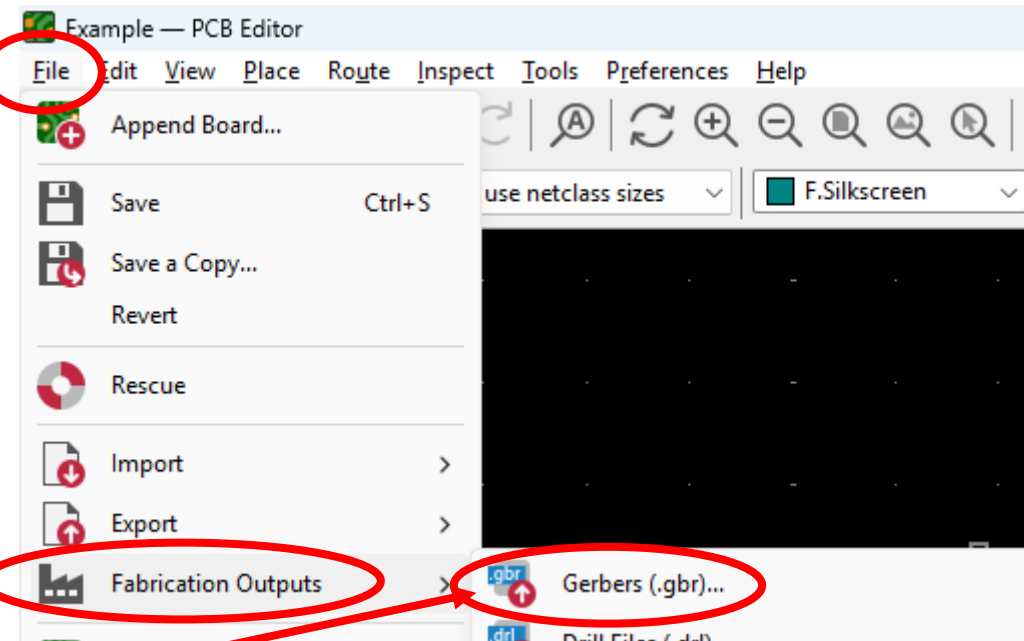
- Generate Gerbers and Drill Files
- Manufacturers Each Have Their Own Requirements
 - JLCPCB -- <https://jlcpcb.com/help/article/how-to-generate-gerber-and-drill-files-in-kicad-8>
 - OSH PARK -- <https://docs.oshpark.com/submitting-orders/preorder-checklist/>

PCB Layout Editor

Click File

Hover Over
Fabrication Outputs...

Click Gerbers...



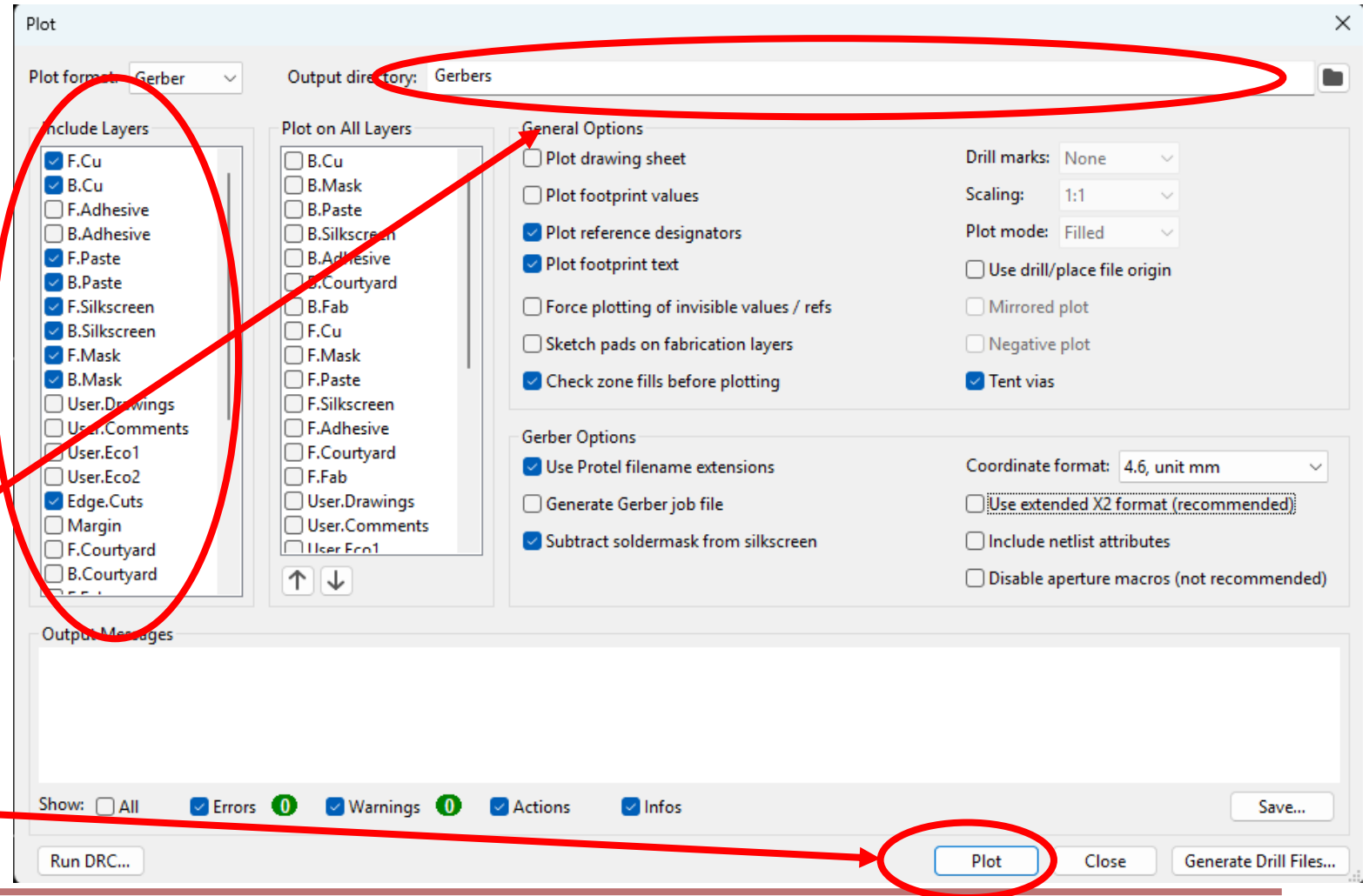
PCB Layout Editor

Included Layers Are Checked

Verify Checkboxes

Add "Gerbers" Directory

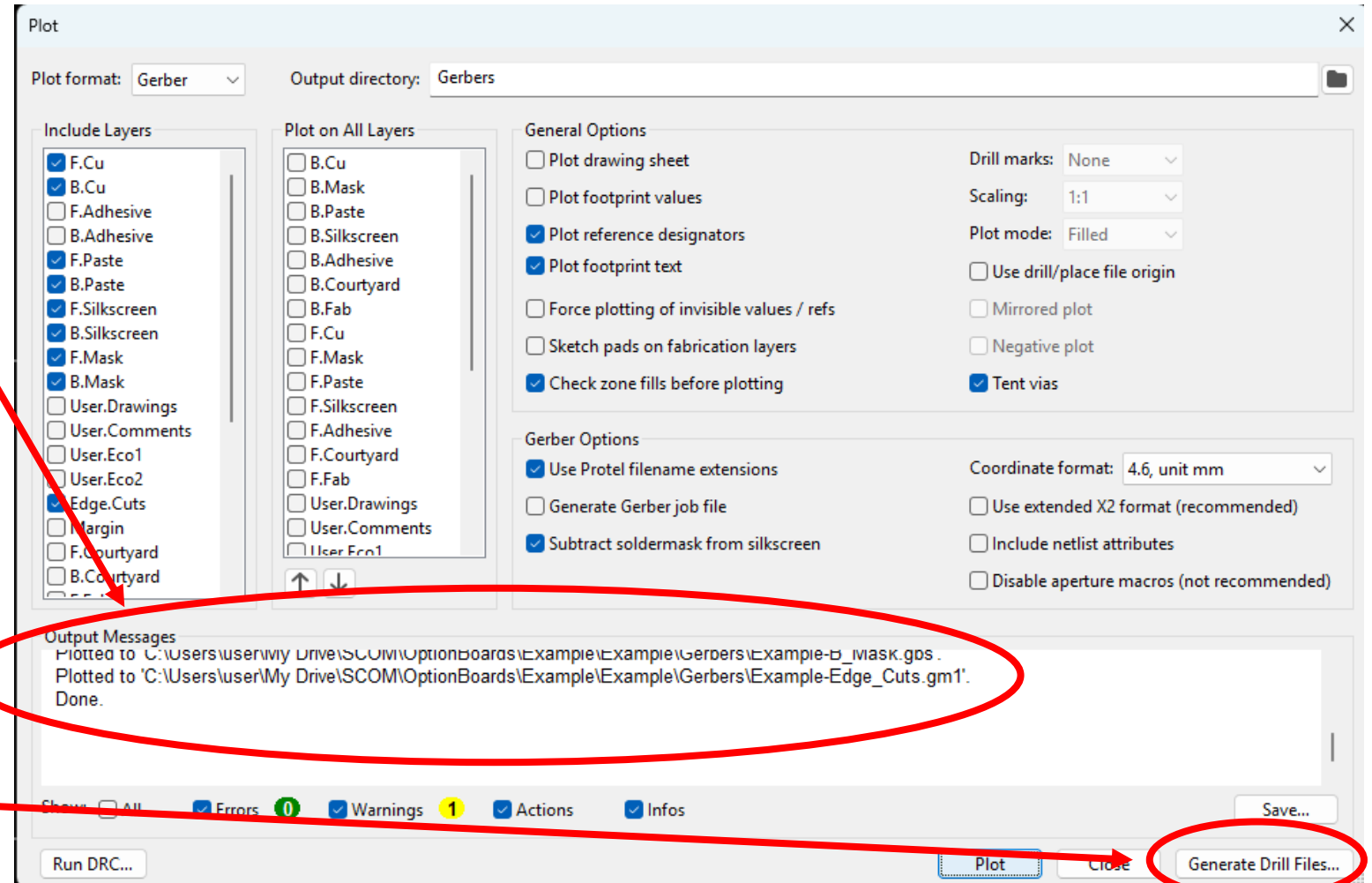
Click Plot



PCB Layout Editor

Output Messages
Verify Files
Generated

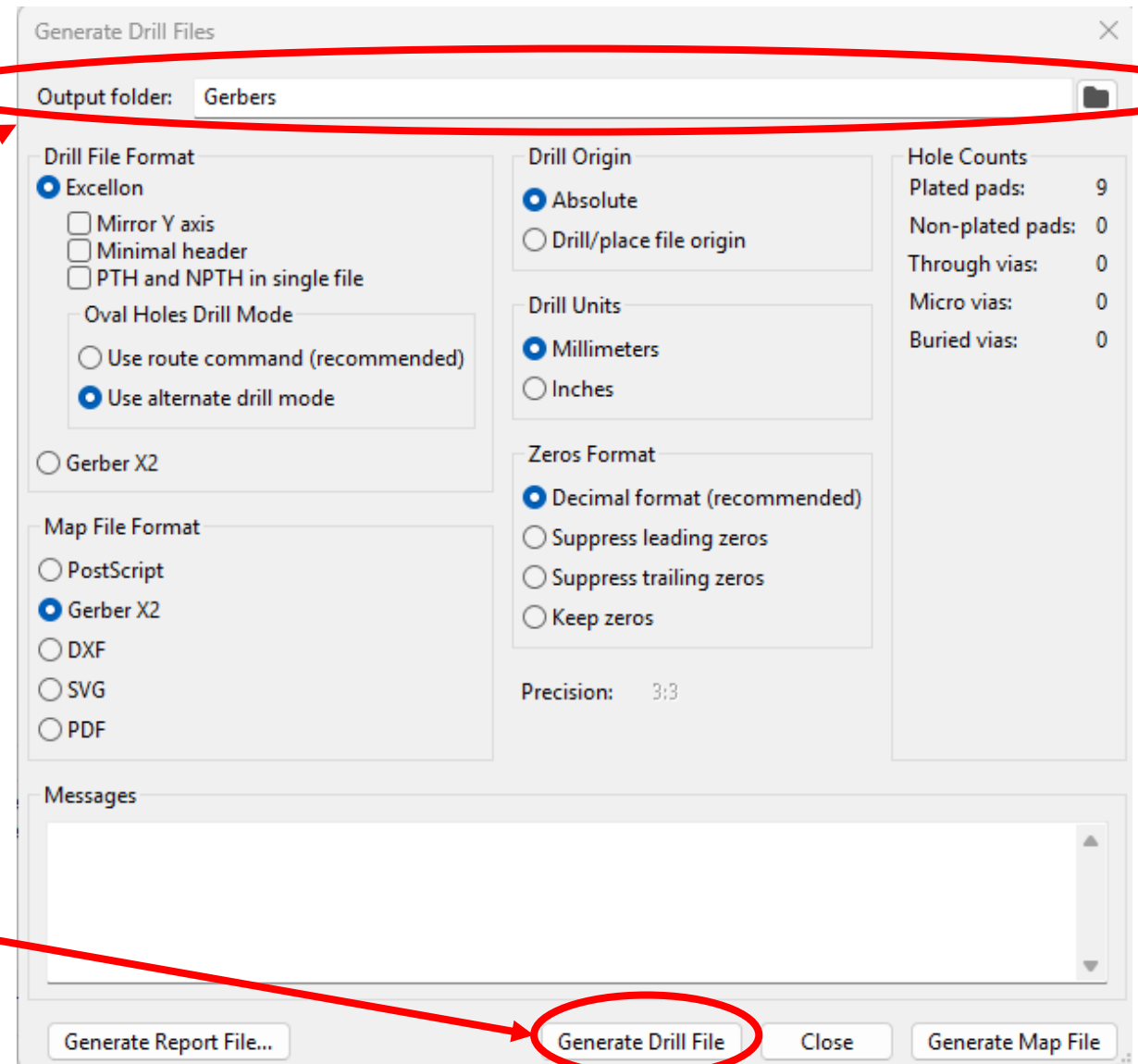
Click Generate
Drill Files



PCB Layout Editor

Default Same Directory
as Gerbers

Click Generate Drill File

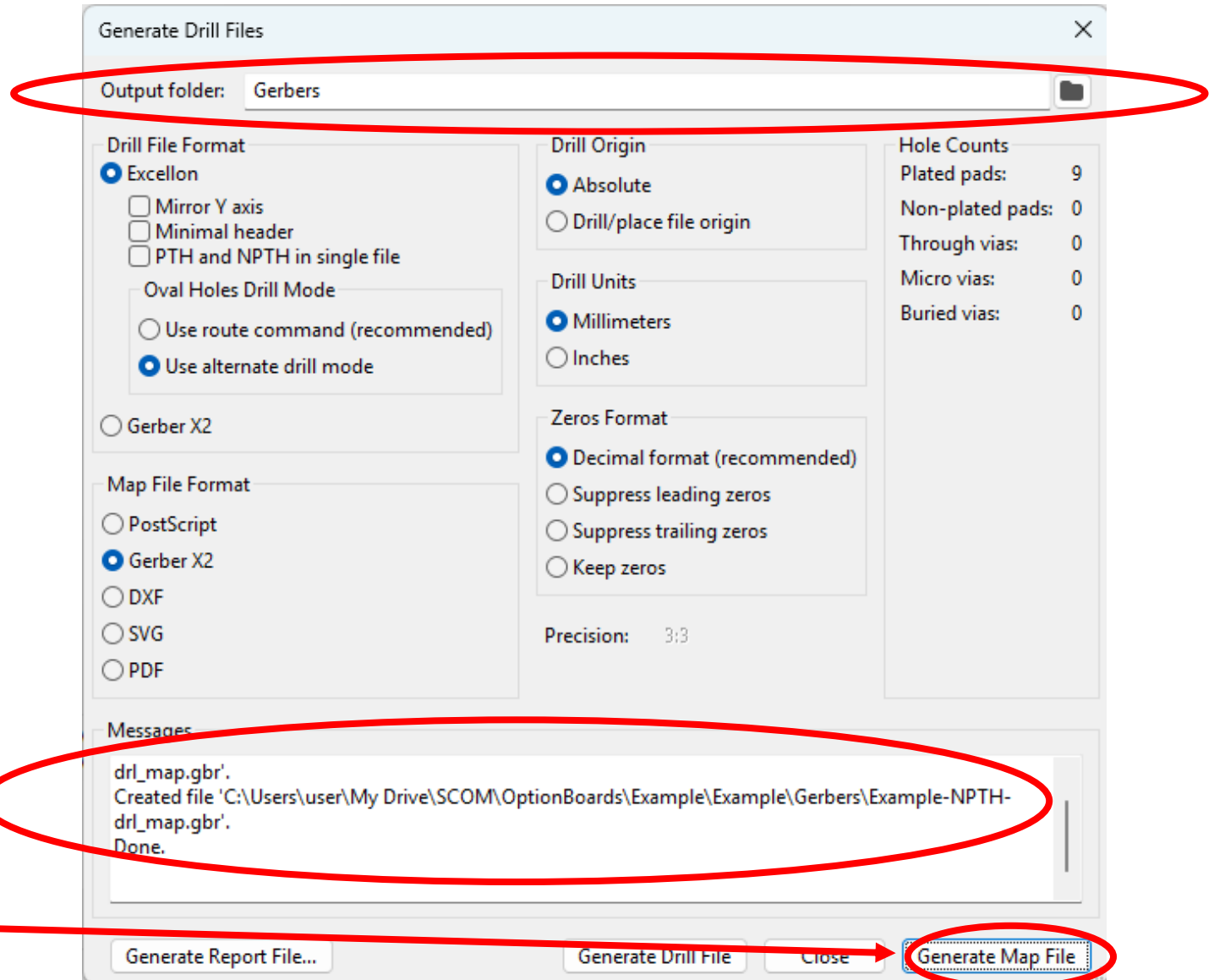


PCB Layout Editor

Default Same Directory
as Gerbers

See Messages

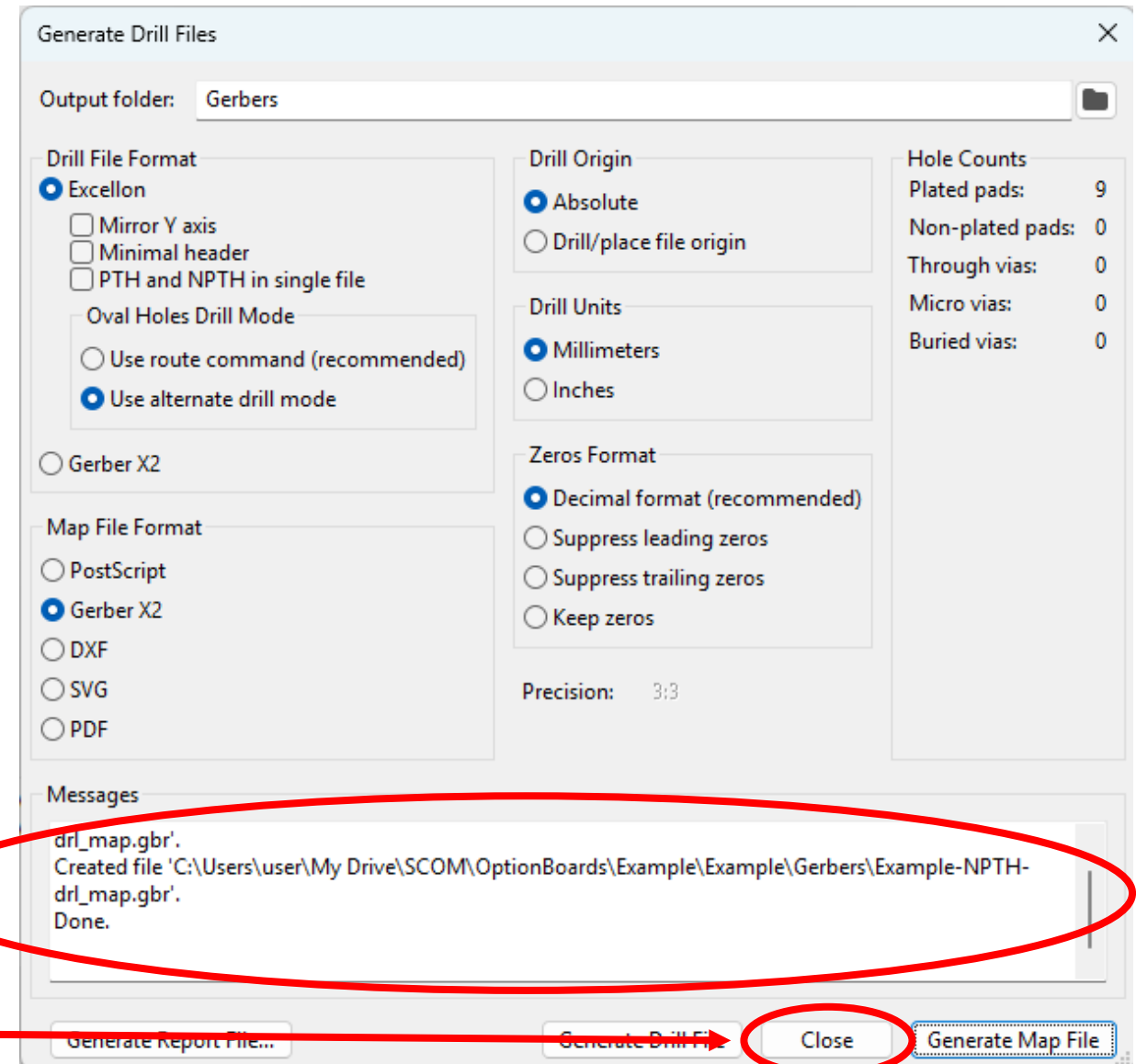
Click Generate Map File



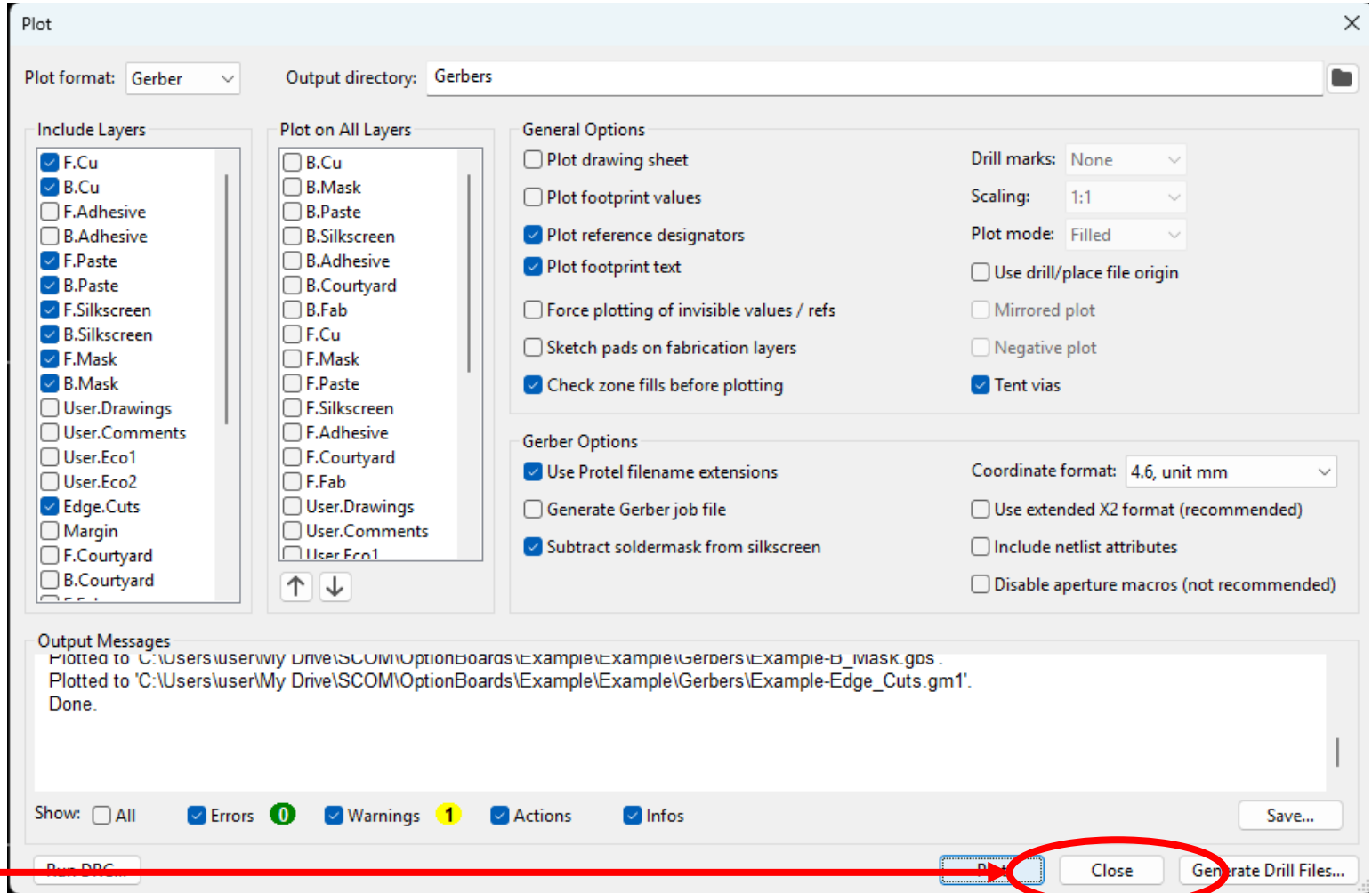
PCB Layout Editor

See Messages

Click Close



PCB Layout Editor

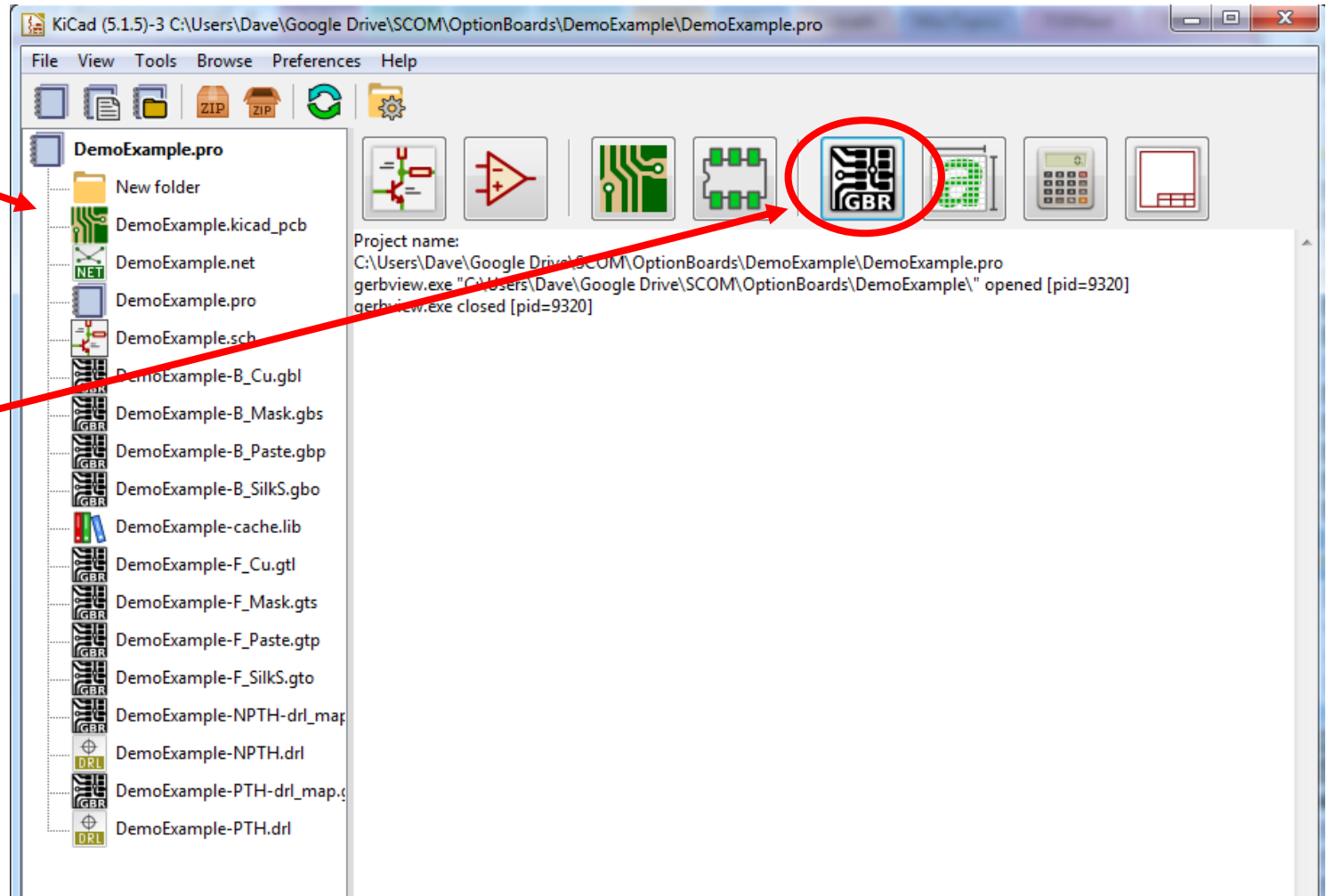


Click Close

KiCad – Main

Output Files

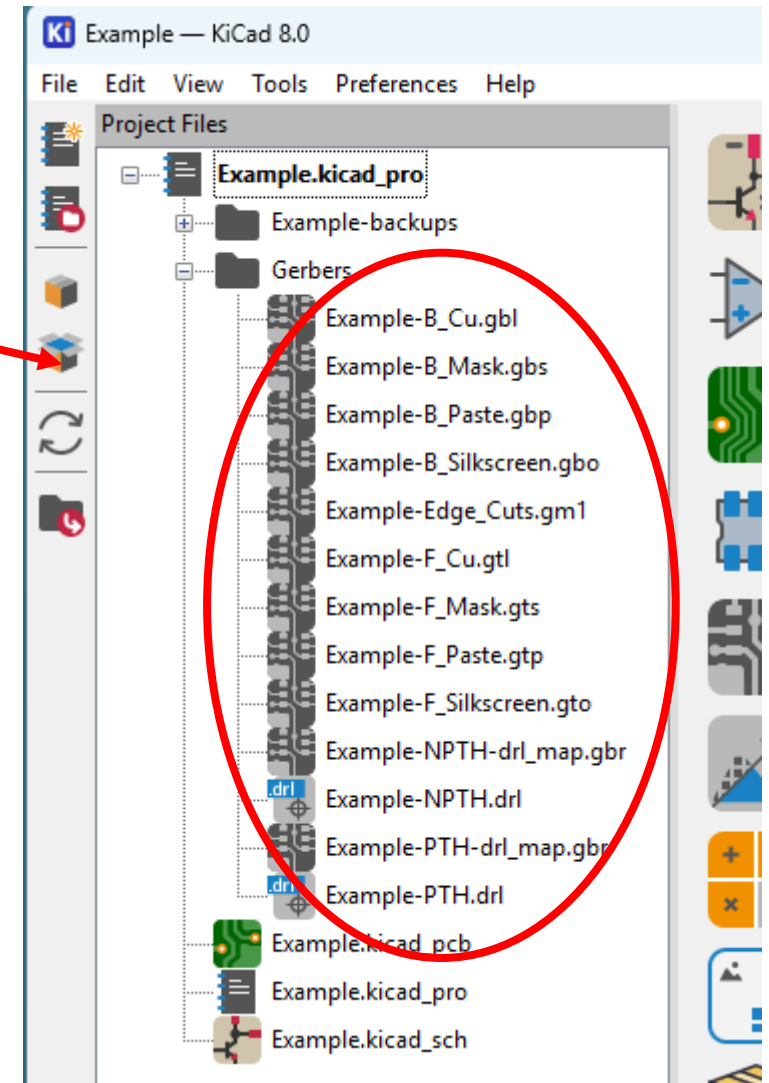
Gerber Viewer
Looks at Output
Files



Zip Gerbers and Drill

Select the Files and Zip them Up

Example_Gerbers_251017.zip



Order Your Boards!!!

Browse to jlcpcb.com

Sign In/Create Account

Upload your .zip file



The screenshot shows the JLCPCB upload interface. A red circle highlights the 'Add gerber file' button, which has an upload icon. A red arrow points from the text 'Upload your .zip file' to this button. To the right of the button is a vertical line with 'OR' text. Further right are three sections: 'Layers' with buttons for 1, 2 (selected), 4, and 6; 'Dimensions' with a box containing '100 x 100 mm'; and 'Quantity' with a box containing '5'. A blue 'Instant Quote' button is on the far right.

Layers	Dimensions	Quantity
1	100 x 100 mm	5
2		
4		
6		

Review Quote.

Order!

References

- KiCad Website
 - <https://www.kicad.org/>
- KiCad Tutorials – Watch for your KiCad Version
 - Lots of tutorials on <https://www.youtube.com/>
- JLCPCB Reference
 - <https://jlcpcb.com/help/article/how-to-generate-gerber-and-drill-files-in-kicad-8>
- Misc
 - https://en.wikipedia.org/wiki/Gerber_format
- Digital Caliper
 - <https://www.amazon.com/dp/B017KUC6XQ>

Questions?

THANK YOU!!!