KiCad QuickStart

Dave Maciorowski, WA1JHK
wa1jhk@macski.us
12 February 2022

JHK Labs

References

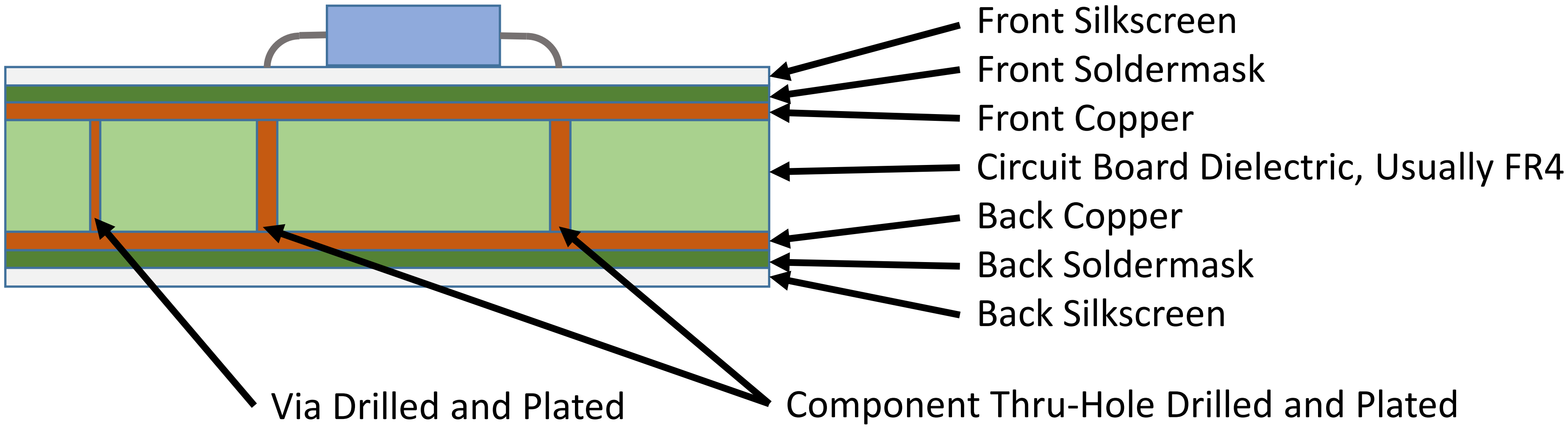
- KiCad Website
  - https://www.kicad.org/
- KiCad Tutorials
  - https://www.youtube.com/c/JohnsBasement
  - https://www.youtube.com/c/contextualelectronics
  - https://learn.sparkfun.com/tutorials/pcb-basics/all
- JLCPCB Reference
  - https://support.jlcpcb.com/article/149-how-to-generate-gerber-and-drill-files-in-kicad
- Misc
  - https://en.wikipedia.org/wiki/Gerber_format

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, Upload to JLCPCB
- References

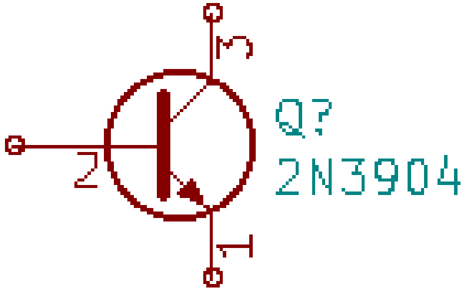
Anatomy of a PCB

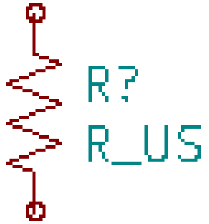
- Many Steps to Manufacturing a PCB
- “Gerber” Files Used in the Manufacturing Process
- KiCad Creates Gerber Files
- “Layers” Refers to the Number of Copper Layers, Always Even, Up to 30

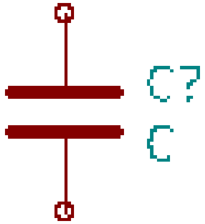


Anatomy of a PCB

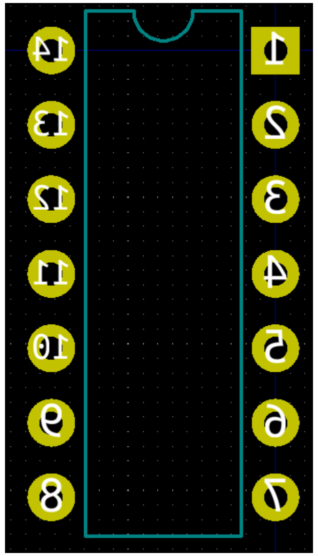
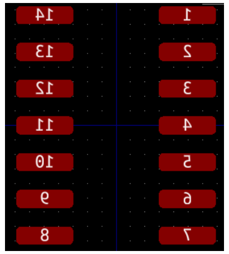
- Schematic Symbols







- Circuit Board Footprints
  - ICs
    - DIP-14
    - SOIC-14

  - Resistors
    - .3 Inch Axial

    - 0603 == .06 x .03 Inch

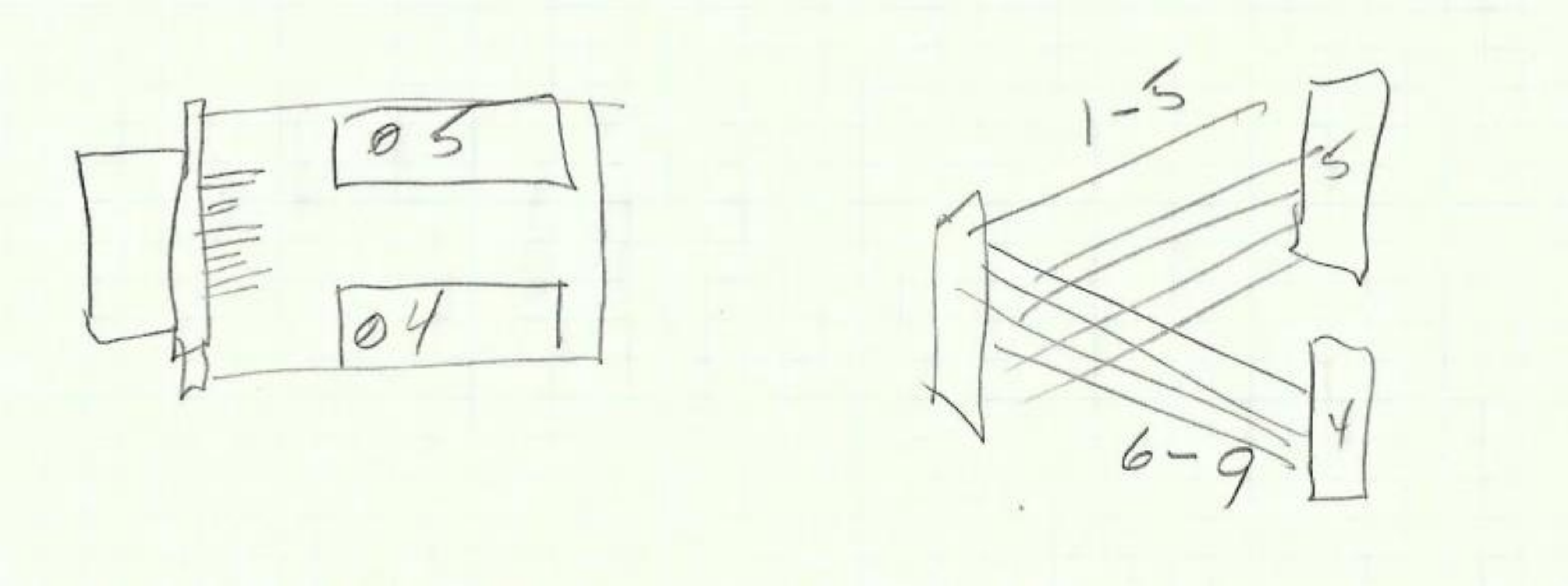
KiCad – Work Flow

- 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
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

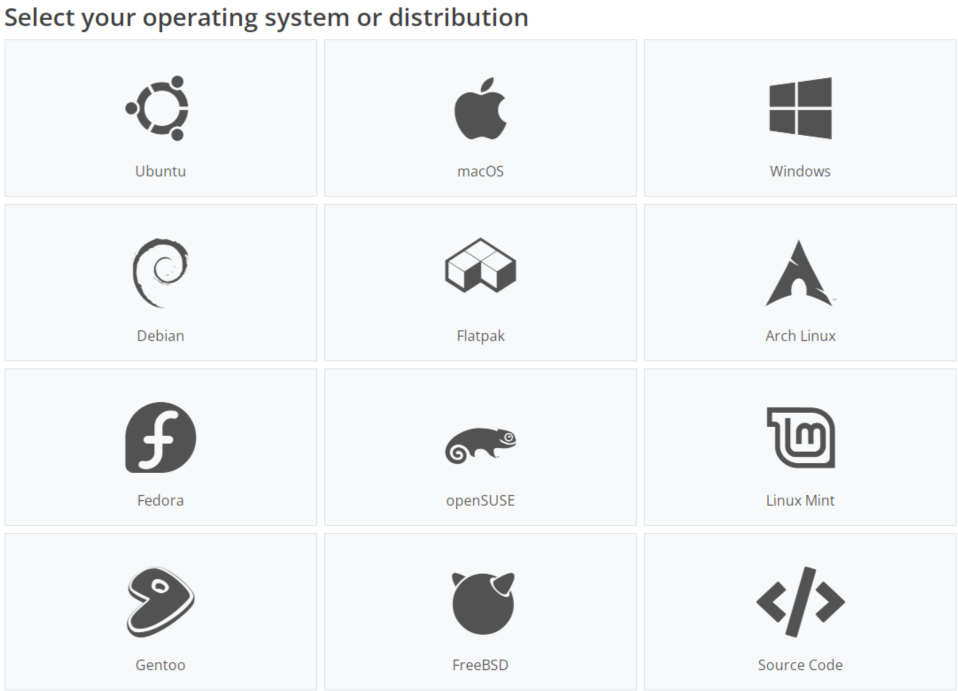
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate Footprint With Component
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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



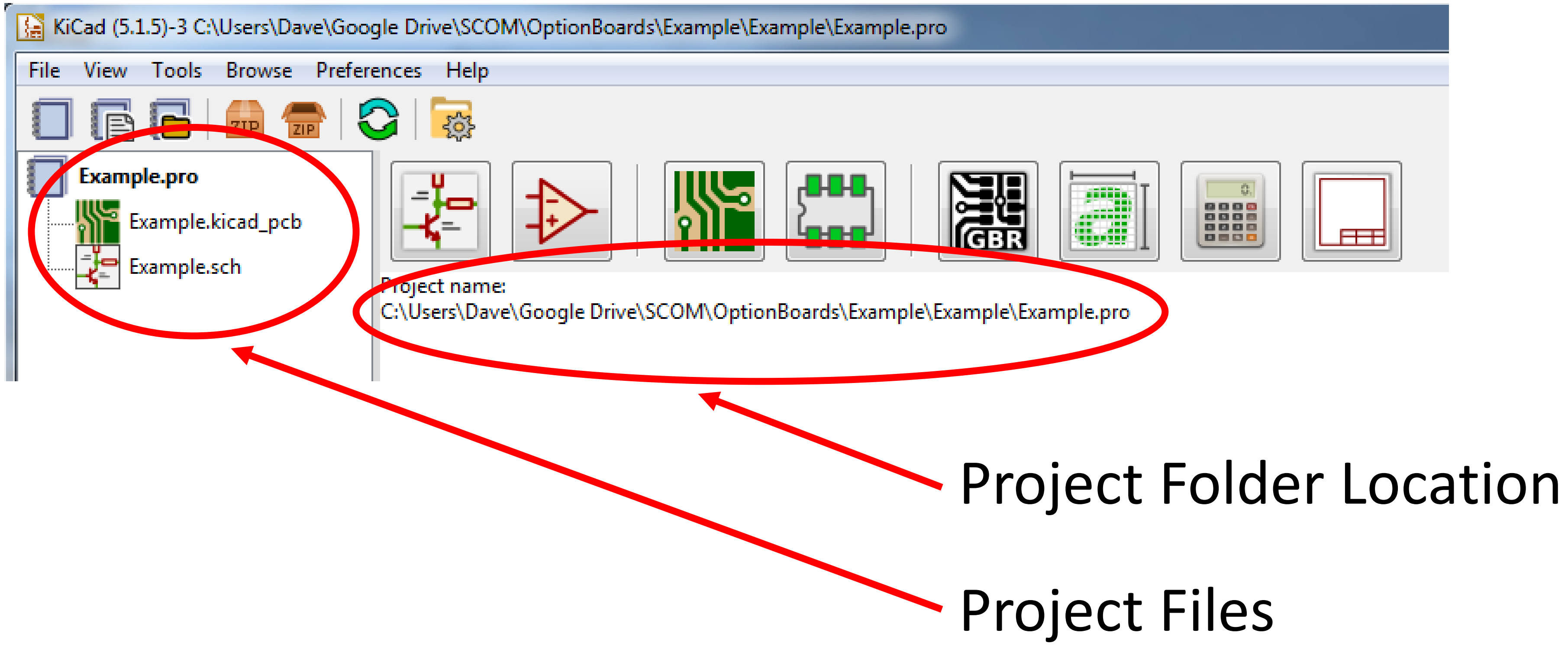
KiCad

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

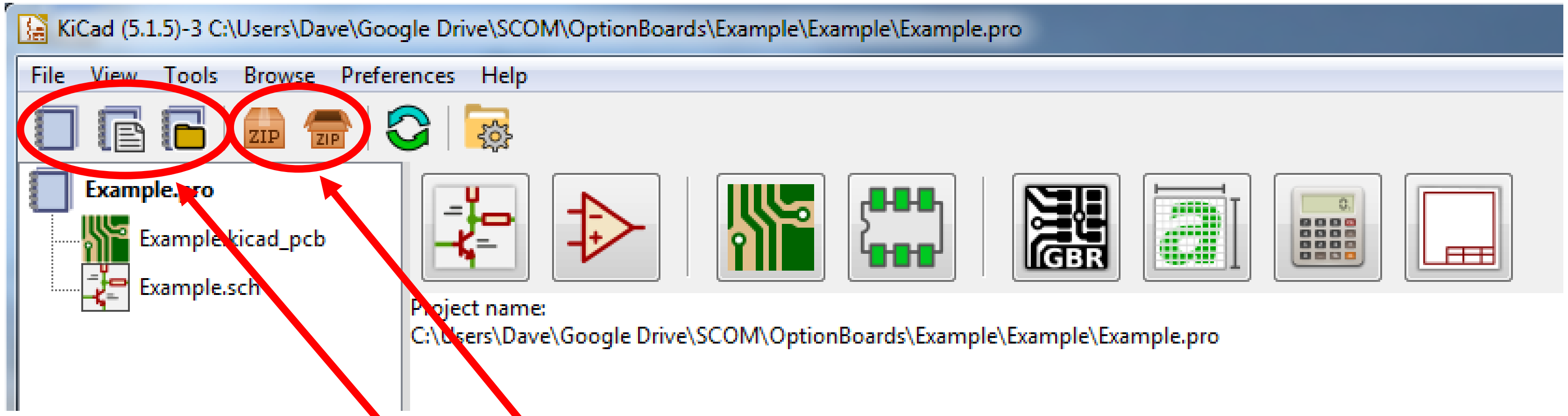


- Choose Your Platform
  - Windows
  - macOS
  - Linux in Many Flavors
  - FreeBSD
  - Source Code

KiCad – Main Screen



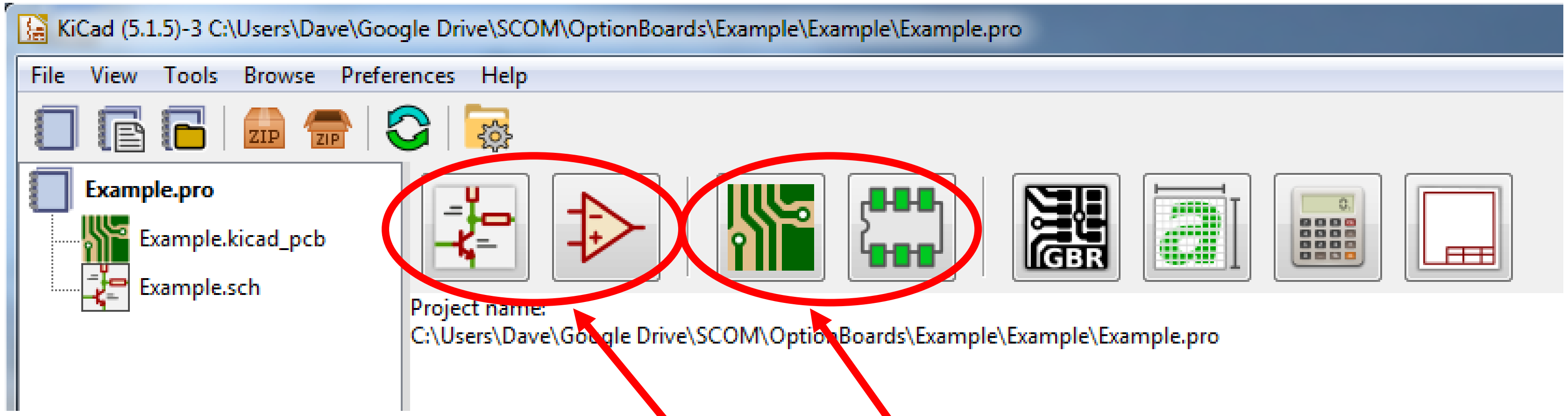
KiCad – Main Screen



Archive and Unarchive Project

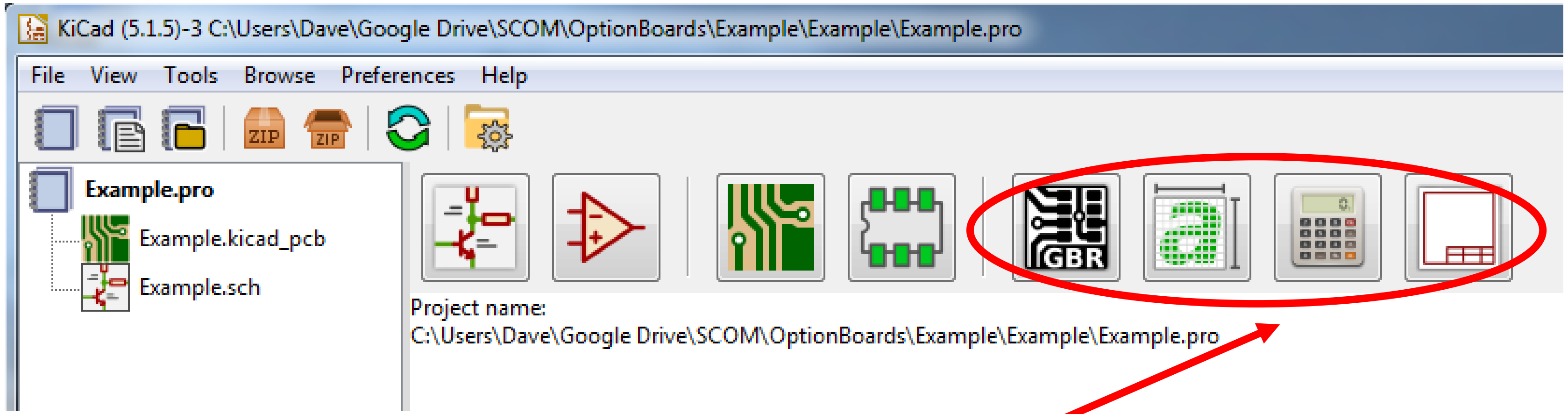
Create and Open Project Files

KiCad – Main Screen



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

KiCad – Main Screen

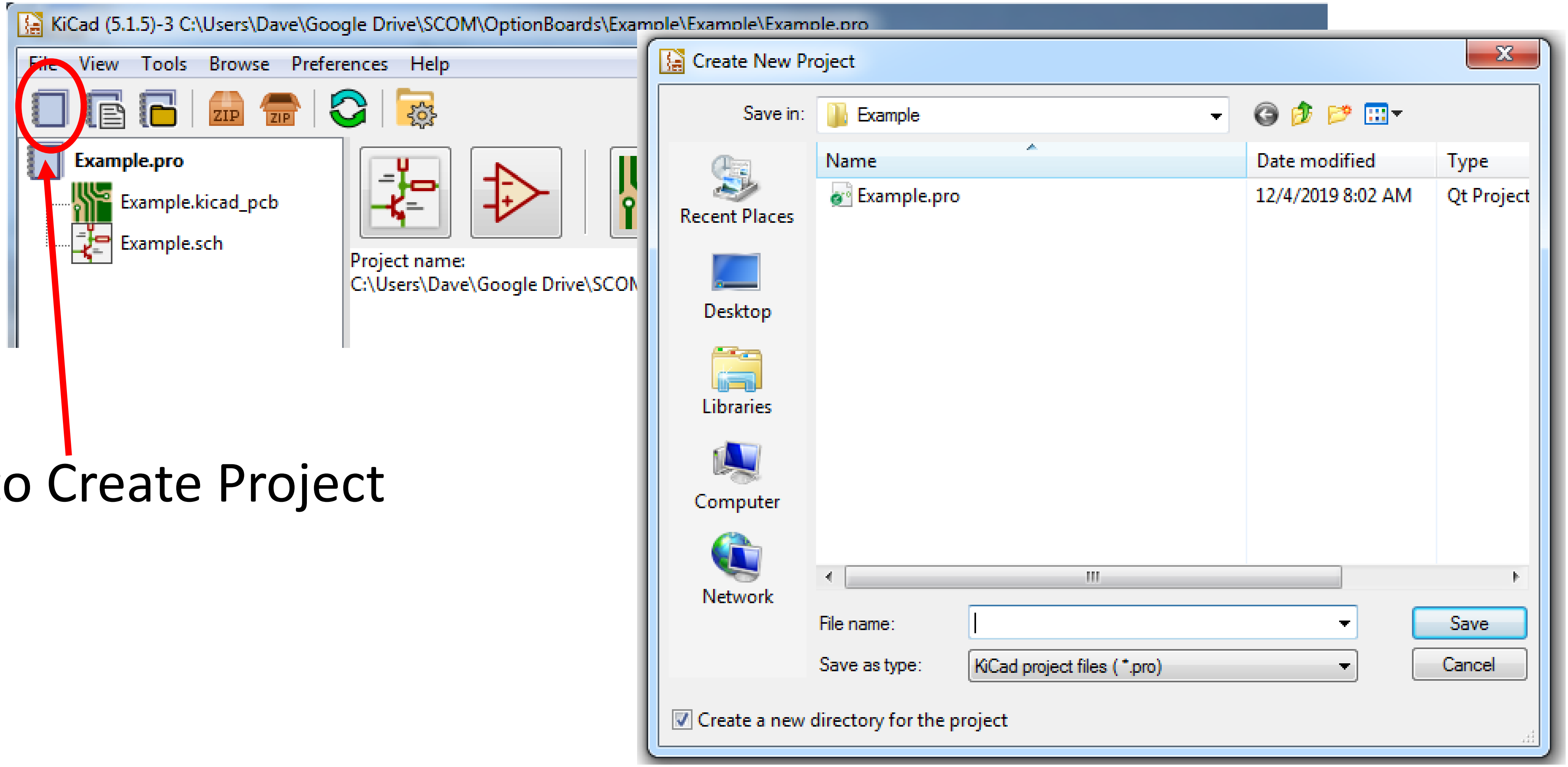


- Gerber Viewer
- Bitmap to Component Converter
- PCB Calculator
- Page Layout Editor

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
  - Import Netlist
  - Mounting Holes
    - 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

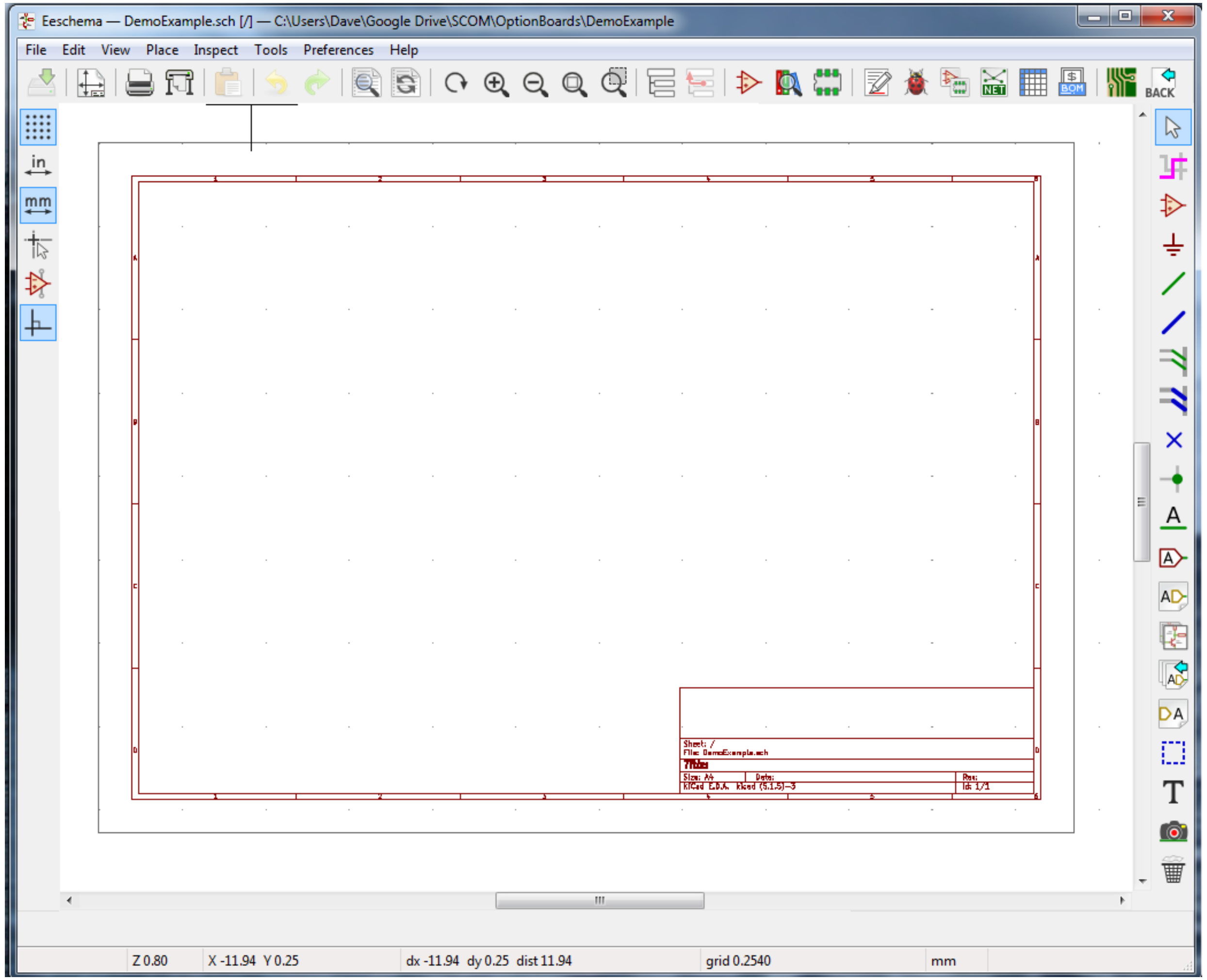


Click to Create Project

KiCad – Work Flow

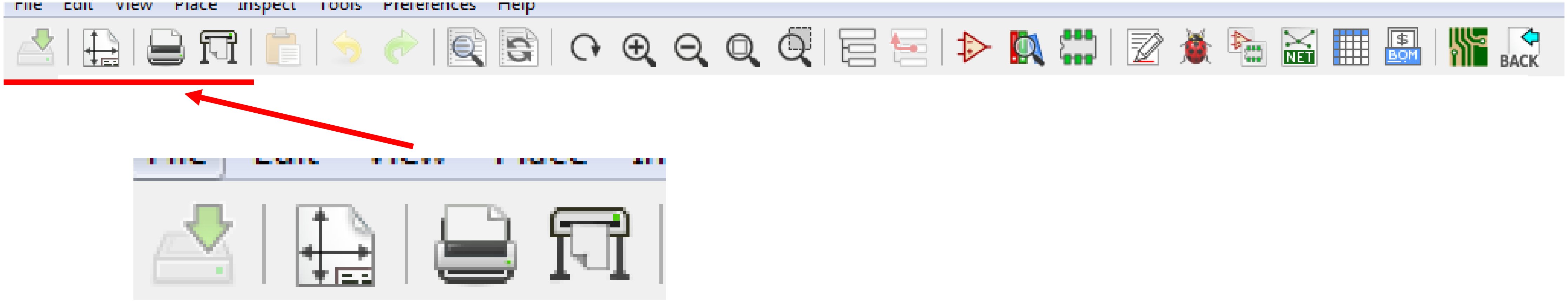
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate Footprint With Component
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

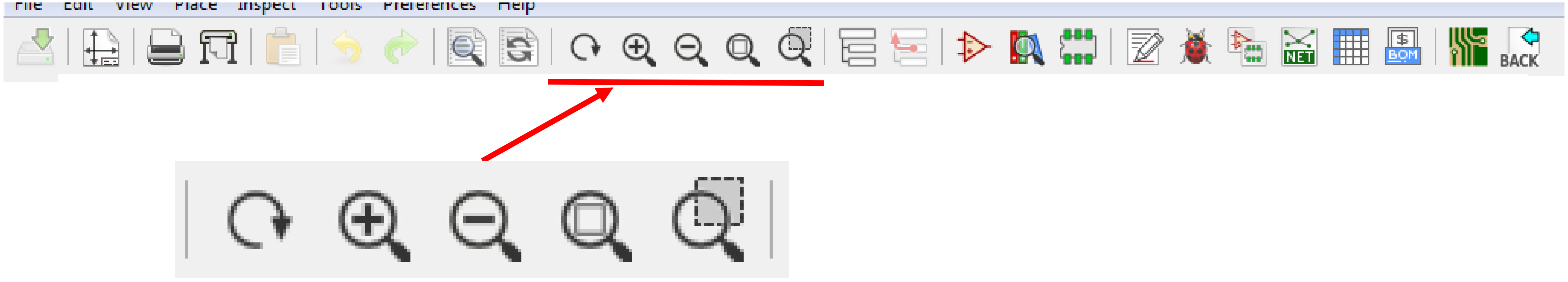


- Save All
- Page Size – Set the schematic sheet size
- Print – Schematic to a Printer
- Plot – Schematic to a PDF

Schematic Layout Editor

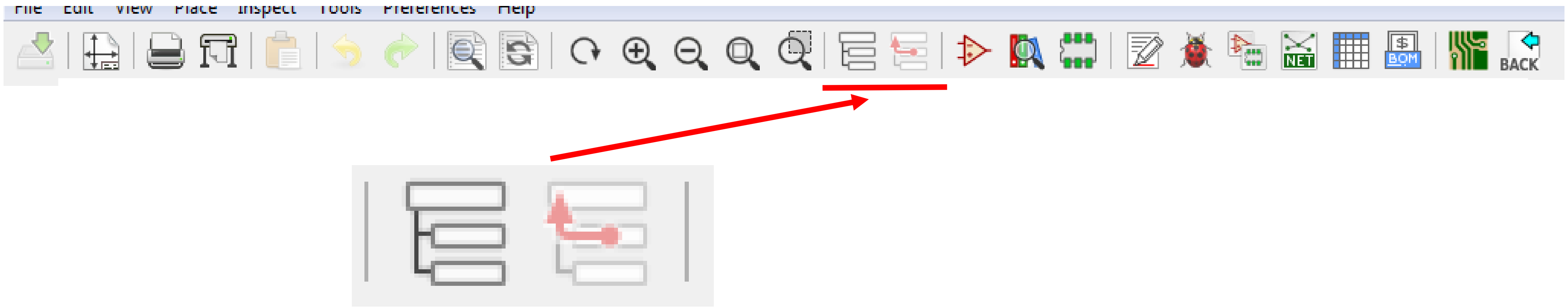
- Editing
  - Paste -- CTRL-V
  - Undo – CTRL-Z
  - Redo – CTRL-Y
  - Find Symbols and Text – CTRL-F
  - Find and Replace Symbols and Text

Schematic Layout Editor



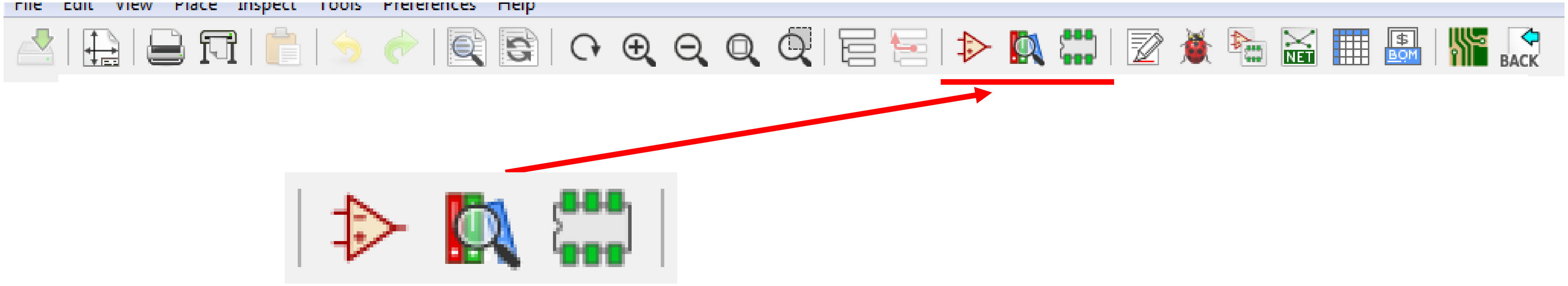
- Schematic View
  - Redraw View – F3
  - Zoom In – F1, also Scroll Wheel
  - Zoom Out – F2 , also Scroll Wheel
  - Zoom to Fit Screen – HOME
  - Zoom to Selection – Mode Button

Schematic Layout Editor



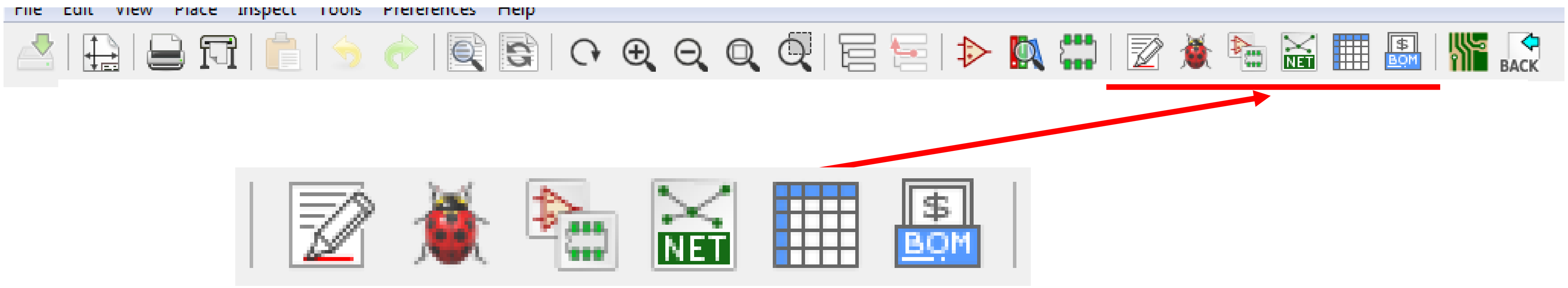
- Multiple Schematic Sheets
  - Navigate Schematic Hierarchy
  - Leave Sheet

Schematic Layout Editor



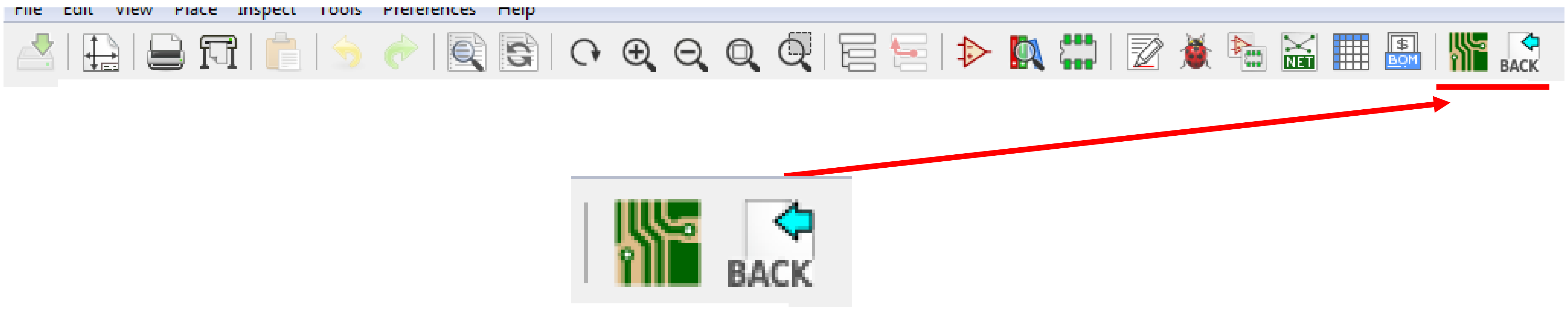
- Symbols and Footprints
  - Create, Delete and Edit Symbols
  - Browse Symbol Libraries – *Spend Some Time Here*
  - Create and Edit Symbols

Schematic Layout Editor



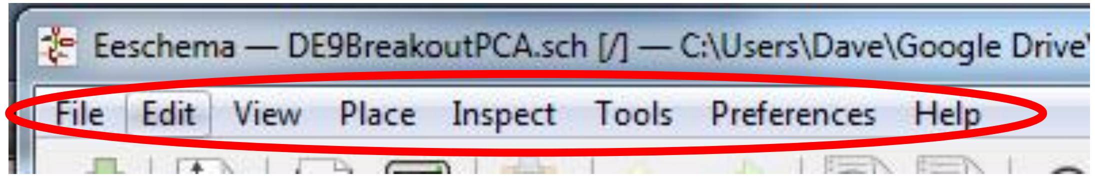
- Process Schematic
  - Annotate Schematic Symbols – Adds Reference Designators
  - Perform Electrical Rules Check – Find Errors that Would Prevent Netlist
  - Assign PCB Footprints to Schematic Symbols – Resolve Errors from Check
  - Generate Netlist – Will be used by PCB Layout Editor
  - Edit Symbol Fields – Resolve Errors from Check
  - Generate Bill of Materials – Parts List

Schematic Layout Editor



- Start the PCB Layout Editor
- Back-Import Footprint Assignment

Schematic Layout Editor

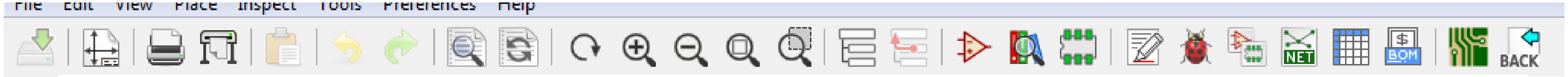


- Menu
  - File Management
  - Editing
  - Alternate Way to Access Tools
  - Preferences
  - Help – List Hotkeys

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
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

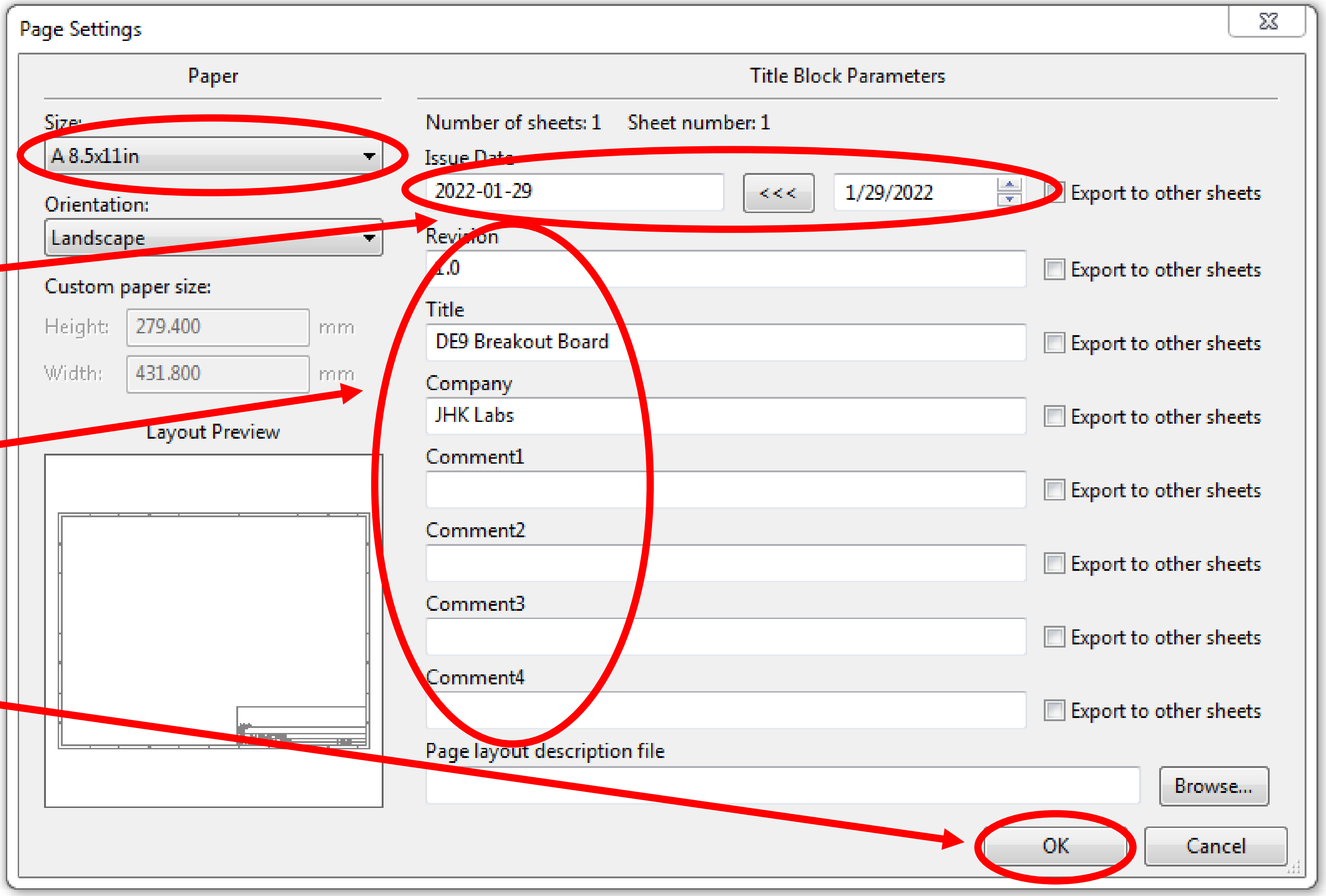
Schematic Layout Editor

Set Paper Size

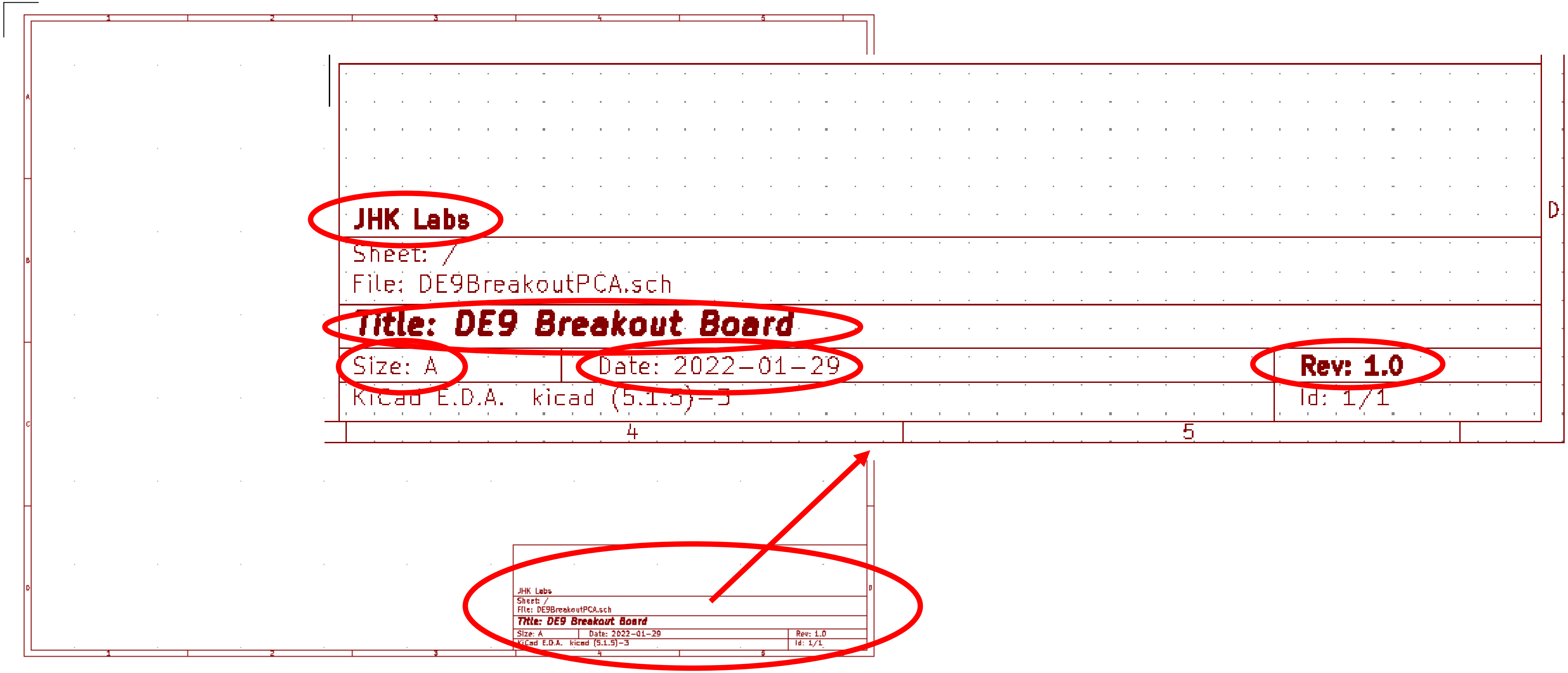
Set Date

Set Title Block Info

Click OK

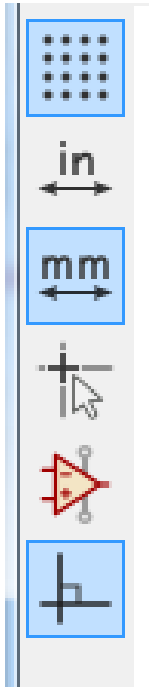


Schematic Layout Editor



Schematic Layout Editor

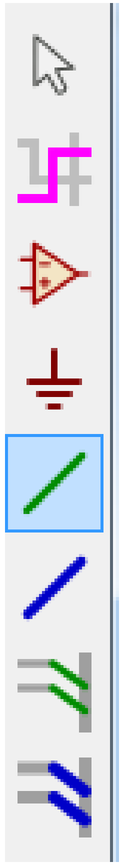
- Left Toolbar – Select Mode



  - Turn the Grid On and Off – Blue is On
  - Switch Units to Inches – 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
  - Toggle Free Angle vs. 90 Degrees

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 Bus to Bus Entry – Click to Attach Bus to Bus

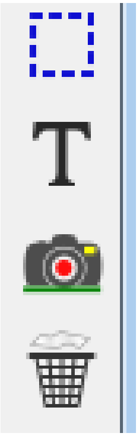
Schematic Layout Editor

- Right Toolbar

  - Place No Connection Flag – Tell Checker it’s ok not to Connect
  - Place Junction – Connect Two Crossed Wires
  - Place Net Label
  - Place Global Label
  - Place Hierarchical Label
  - Create Hierarchical Sheet
  - Place Hierarchical Label Imported from Another Sheet
  - Place Hierarchical Pin in Sheet

Schematic Layout Editor

- Right Toolbar

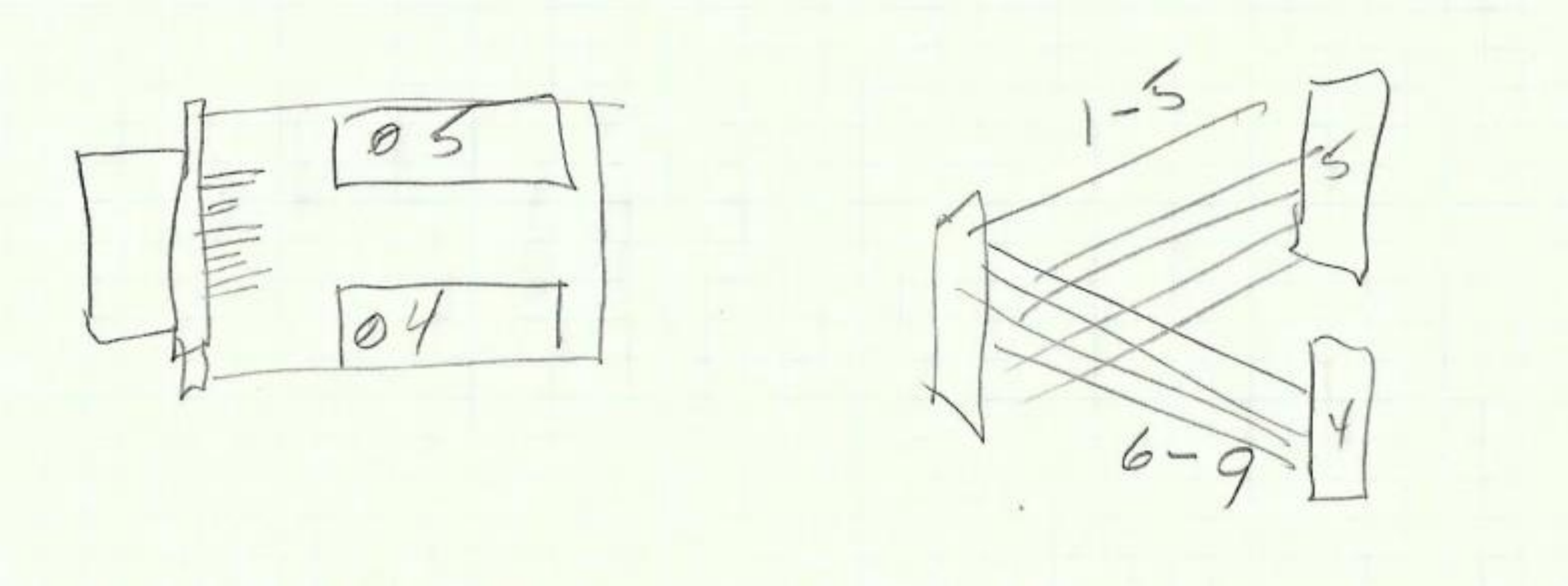


  - Draw Graphic Lines on the Schematic Sheet
  - Place Descriptive Text on the Schematic Sheet
  - Place a Picture on the Schematic Sheet
  - Delete an Item

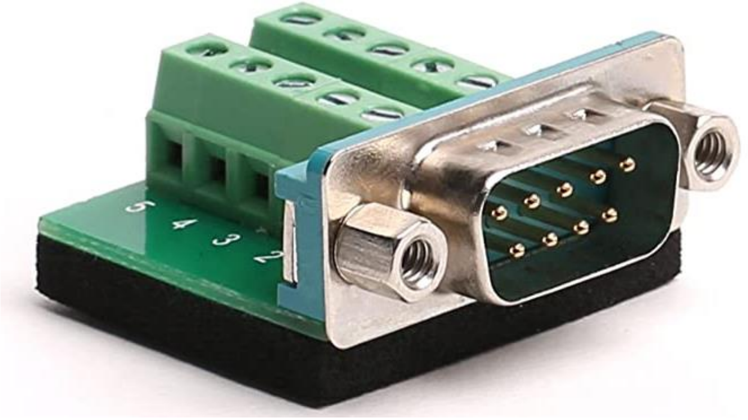
KiCad – Work Flow

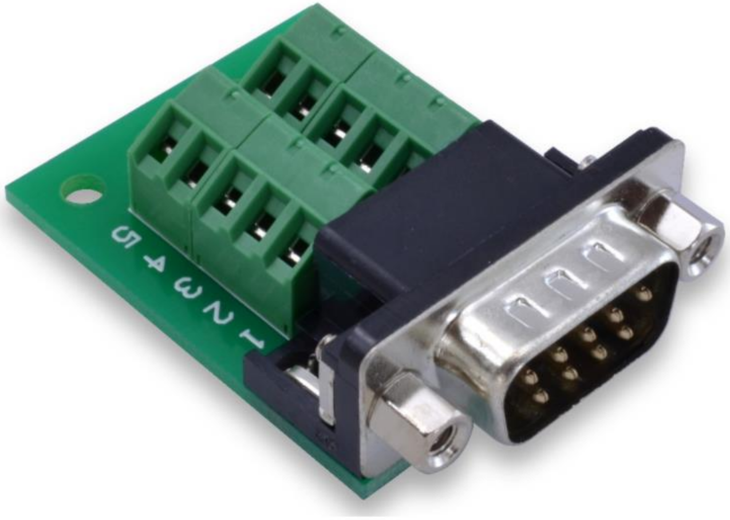
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate Footprint With Component
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

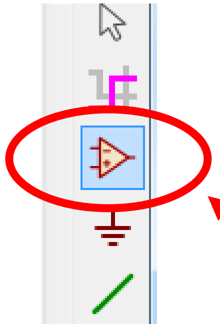


Example Project – Parts



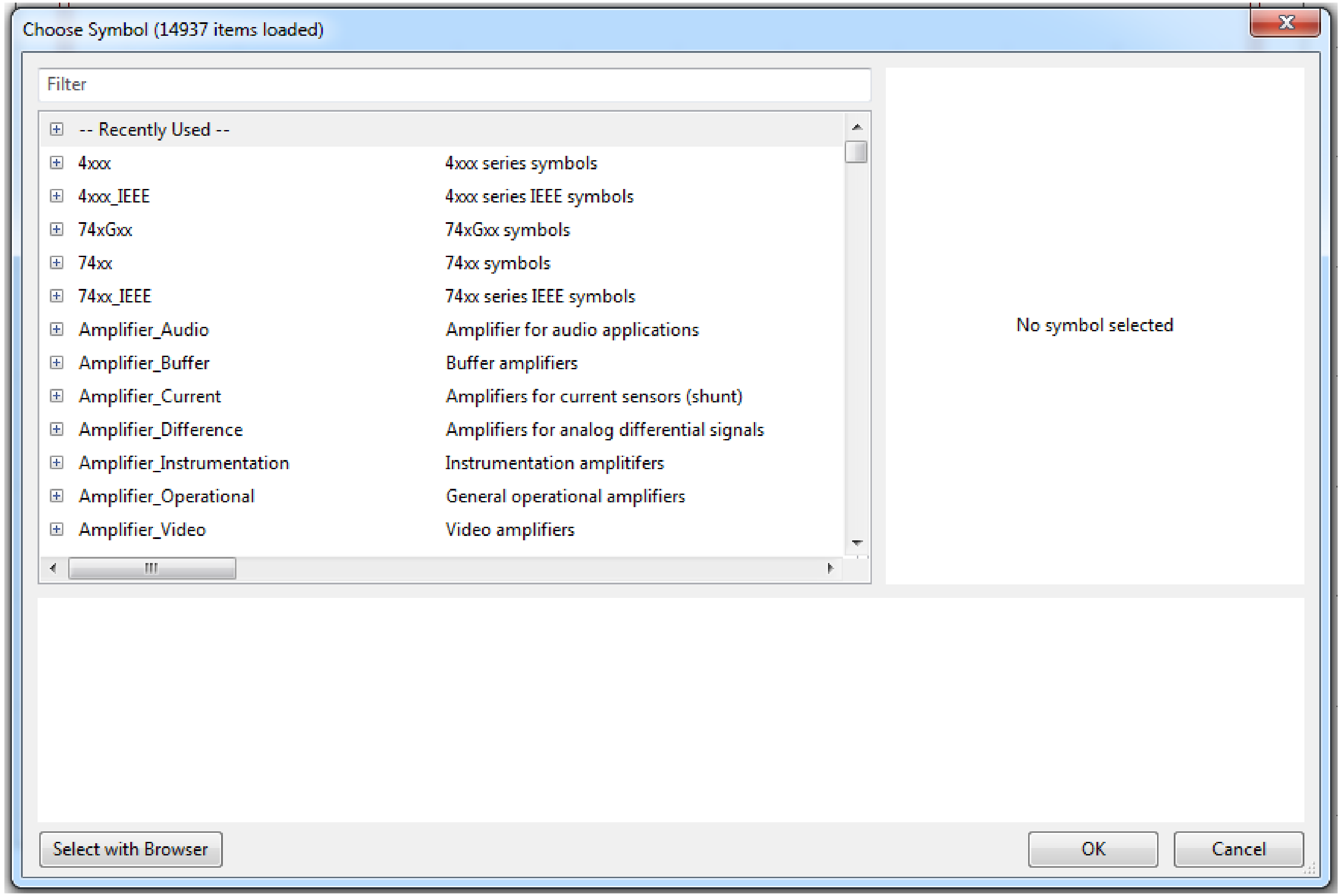


Schematic Layout Editor



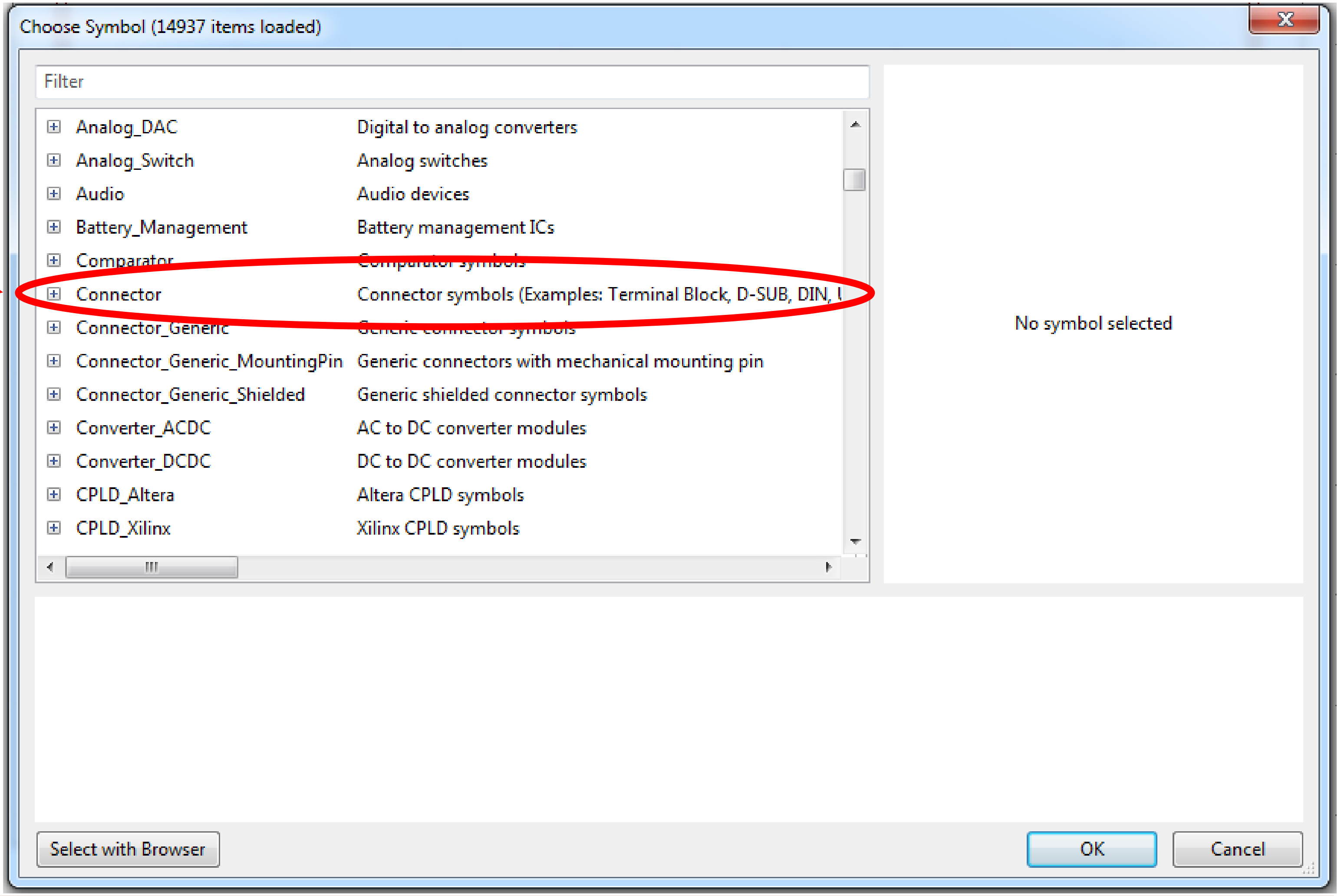
Click Place Symbol then Click Sheet

Symbol Library selection dialog



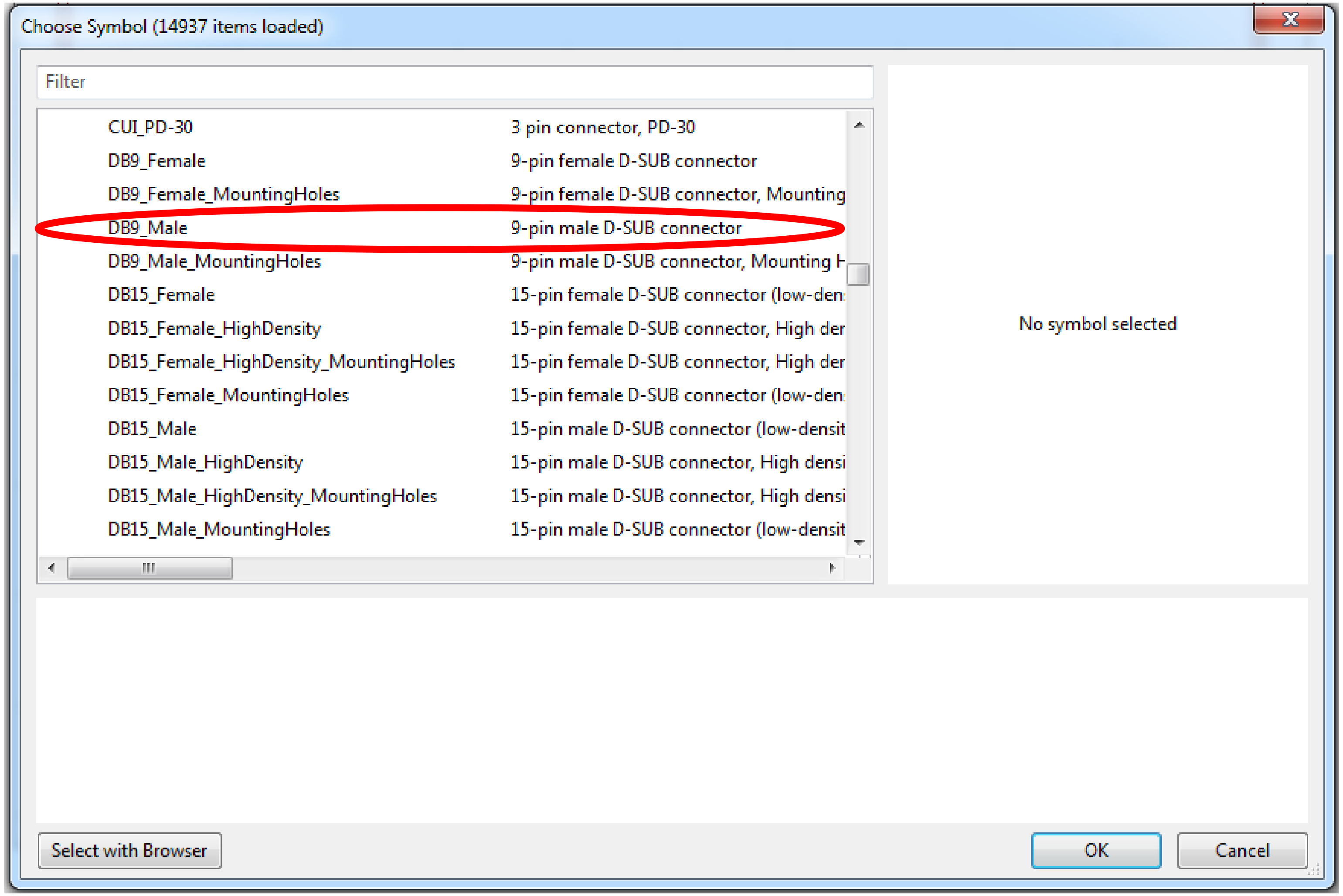
Schematic Layout Editor

Scroll to Connector Category



Schematic Layout Editor

Scroll to DB9_Male Connector

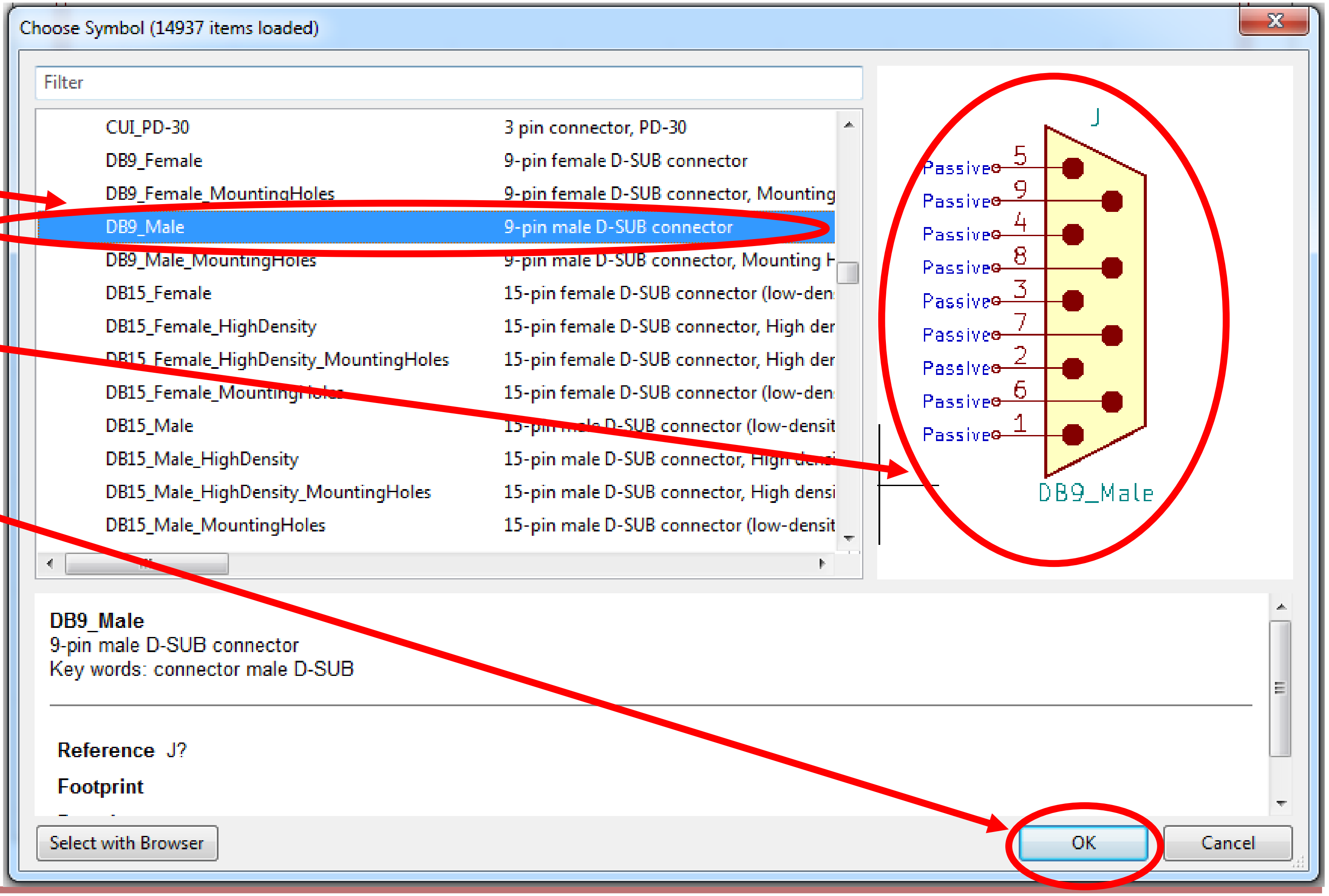


Schematic Layout Editor

Click Symbol Name

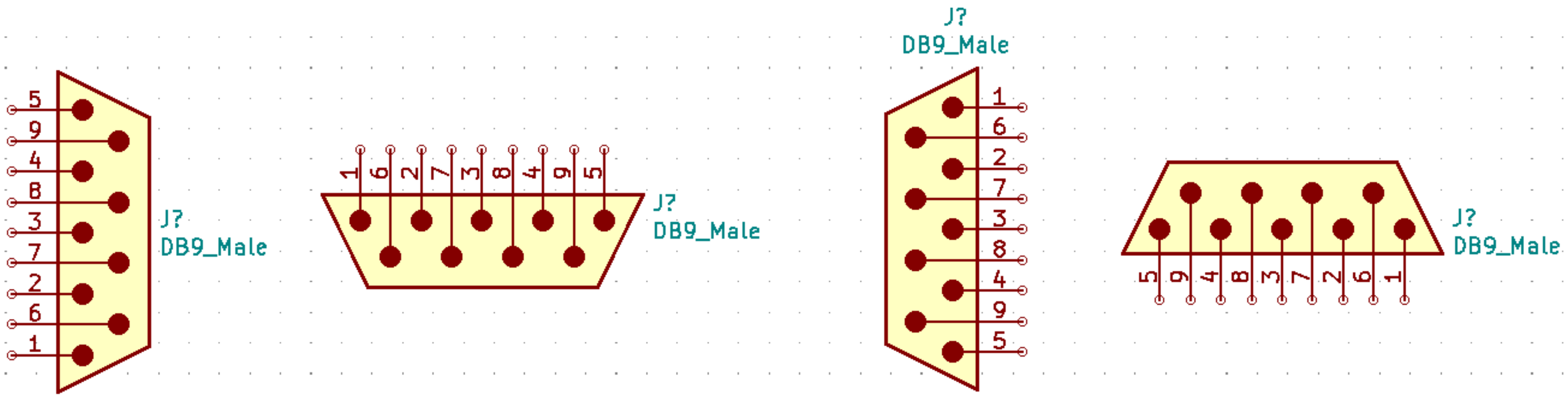
Review Symbol

Click OK to Select



Schematic Layout Editor

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



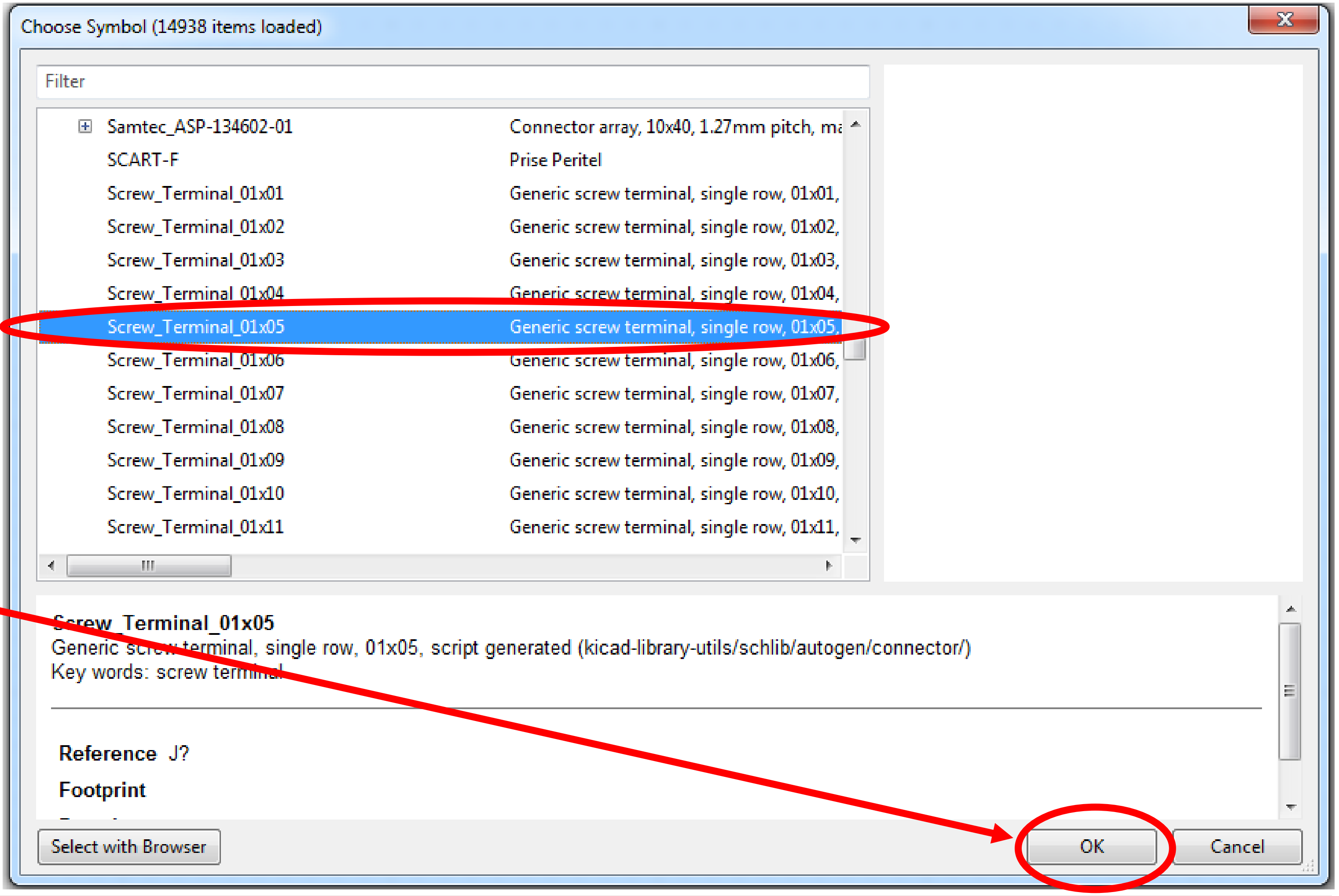
Schematic Layout Editor

Click Sheet

Scroll to Screw_Terminal_01x05

Click OK

Click to Place on Sheet



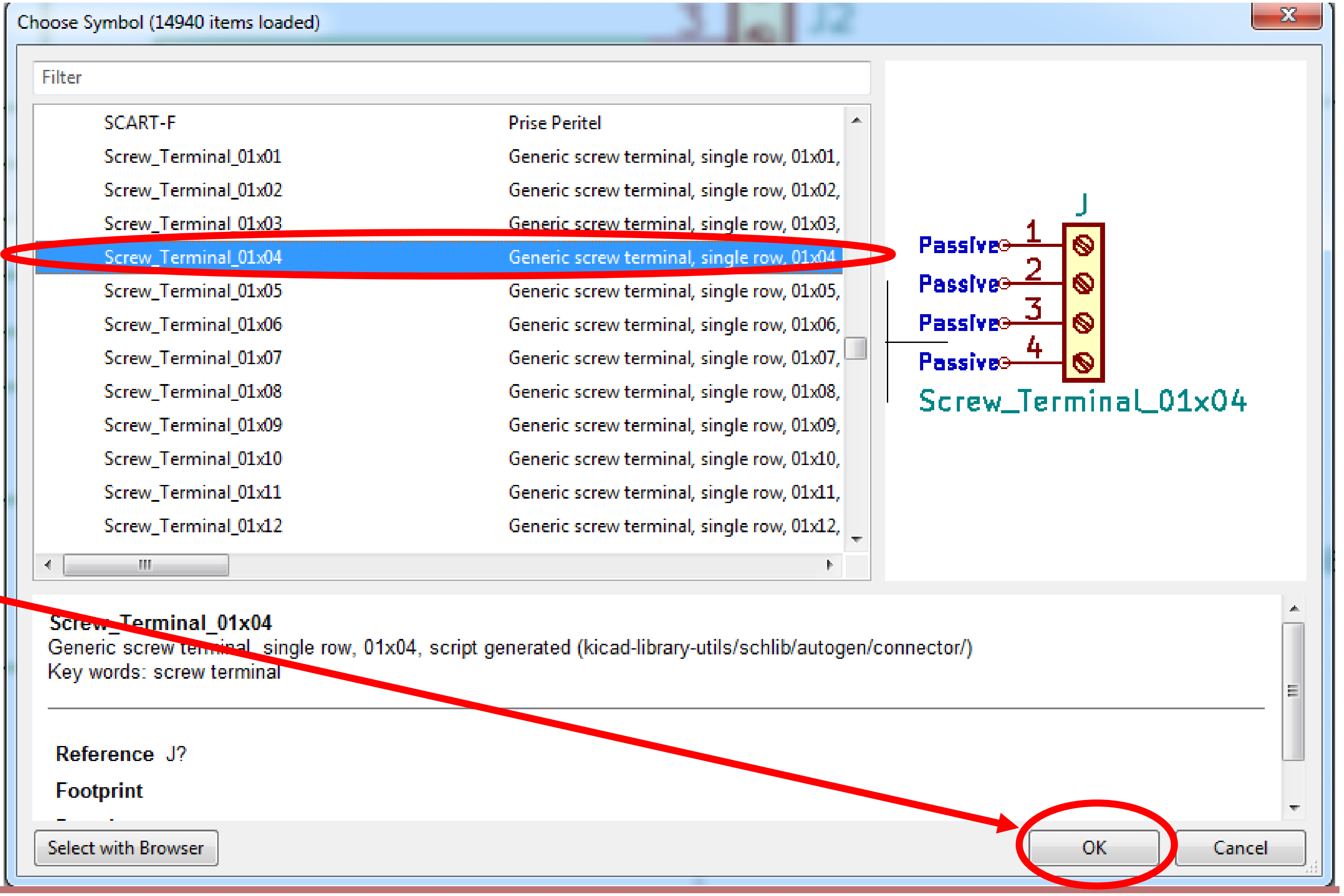
Schematic Layout Editor

Click Sheet

Scroll to Screw_Terminal_01x04

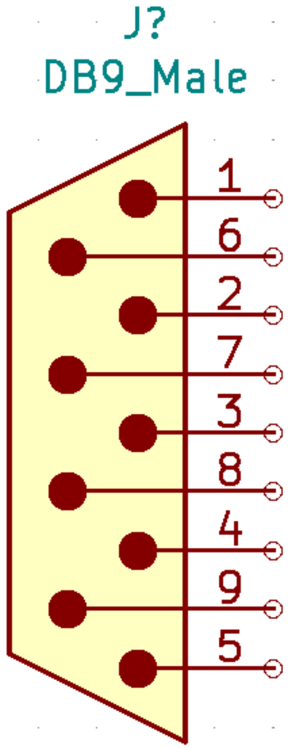
Click OK

Click to Place on Sheet

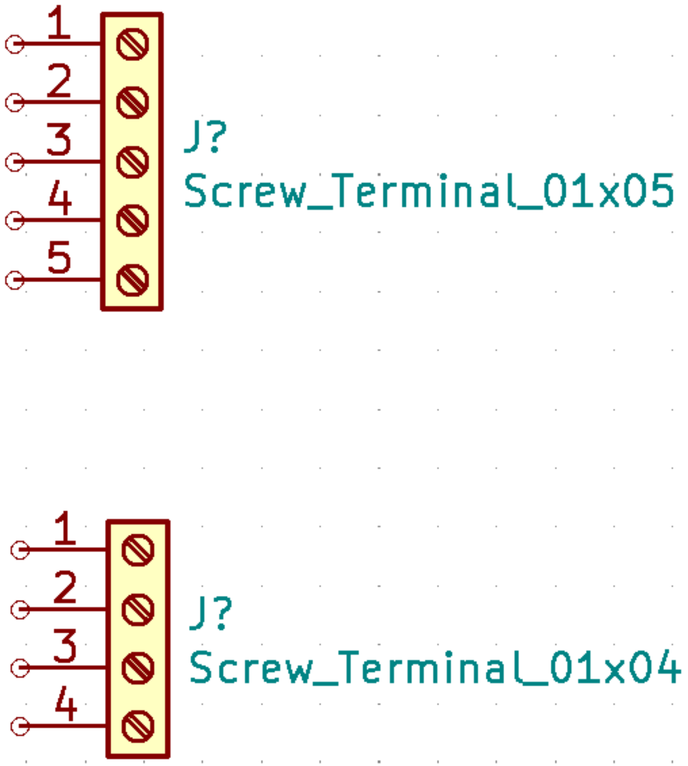


Schematic Layout Editor

Schematic So Far...

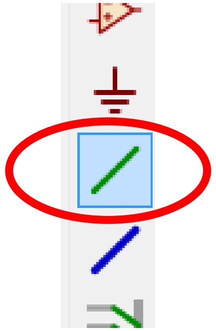


No wires yet...



Schematic Layout Editor

Click Place Wire

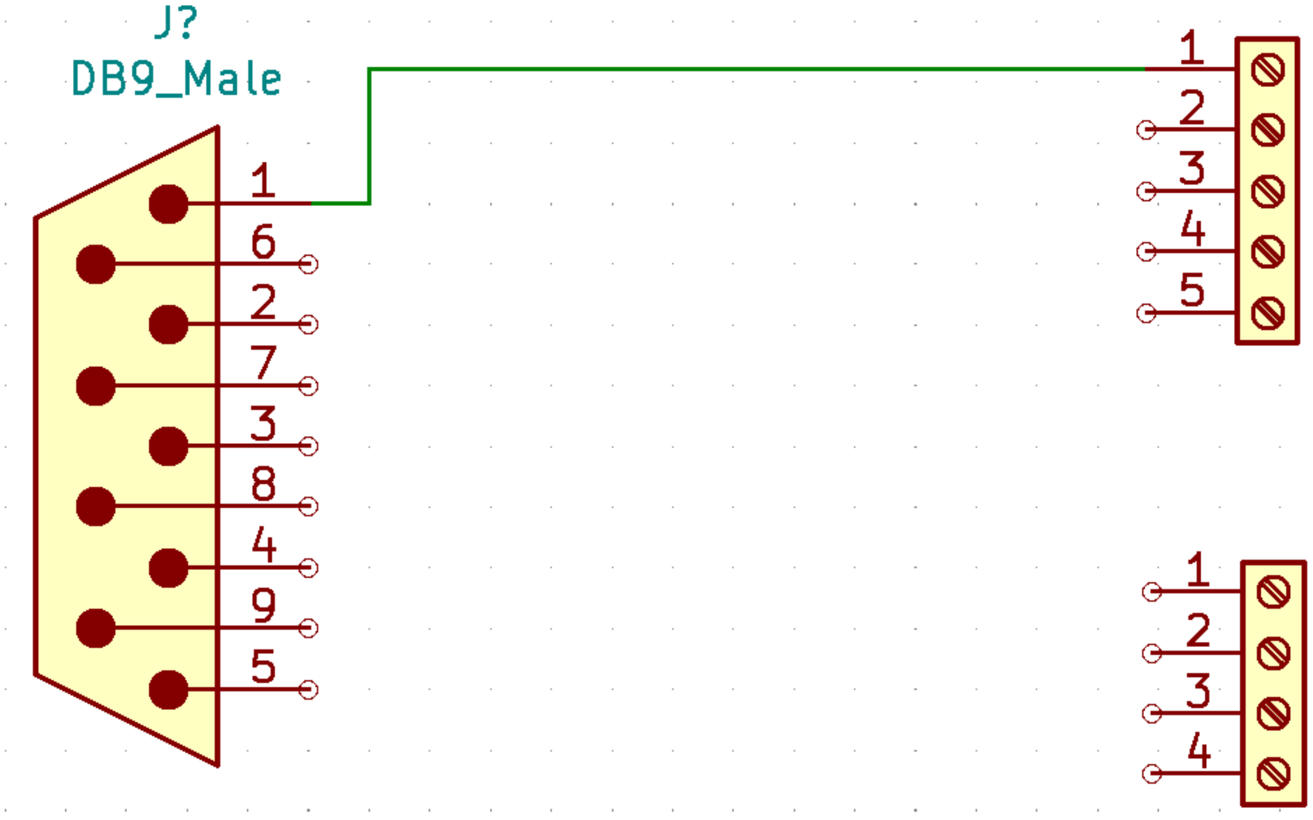


Click Start “Bubble”

Click Corner

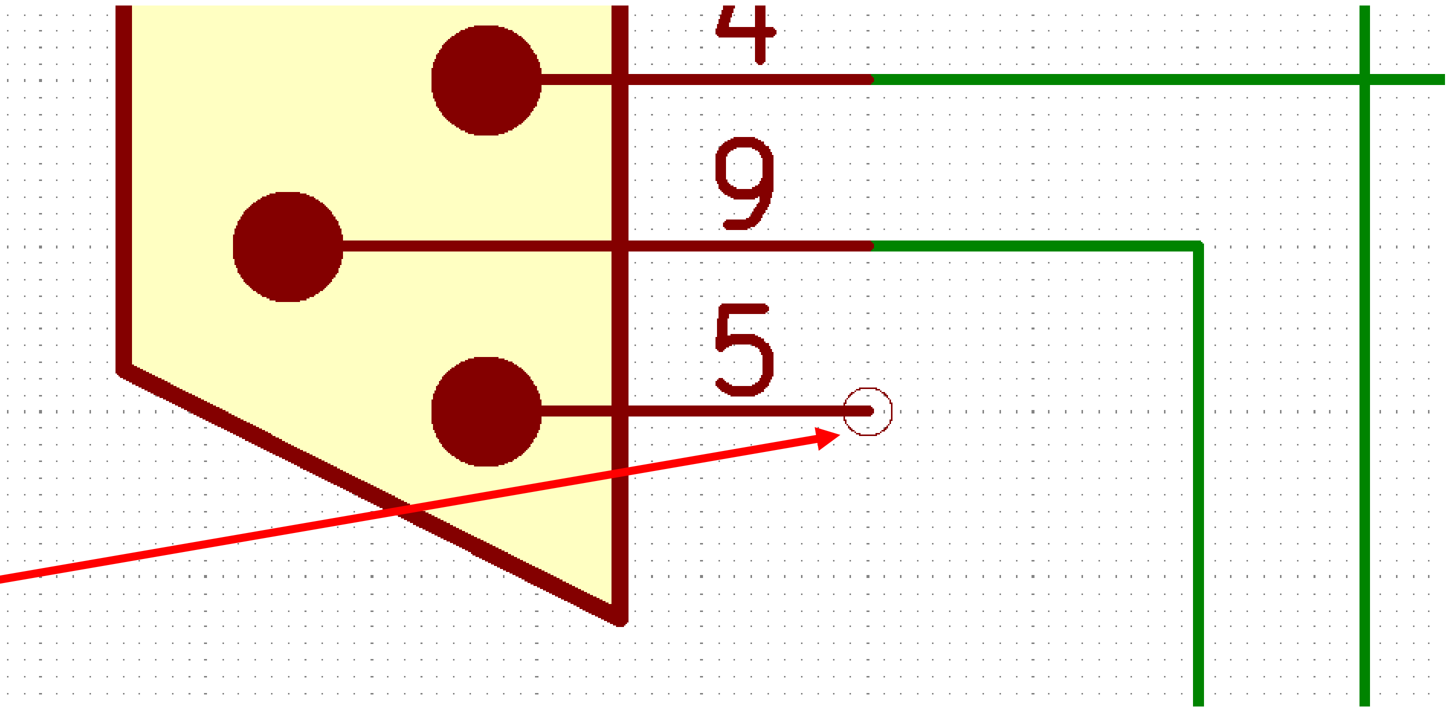
Click End “Bubble”

On Each Wire...



Schematic Layout Editor

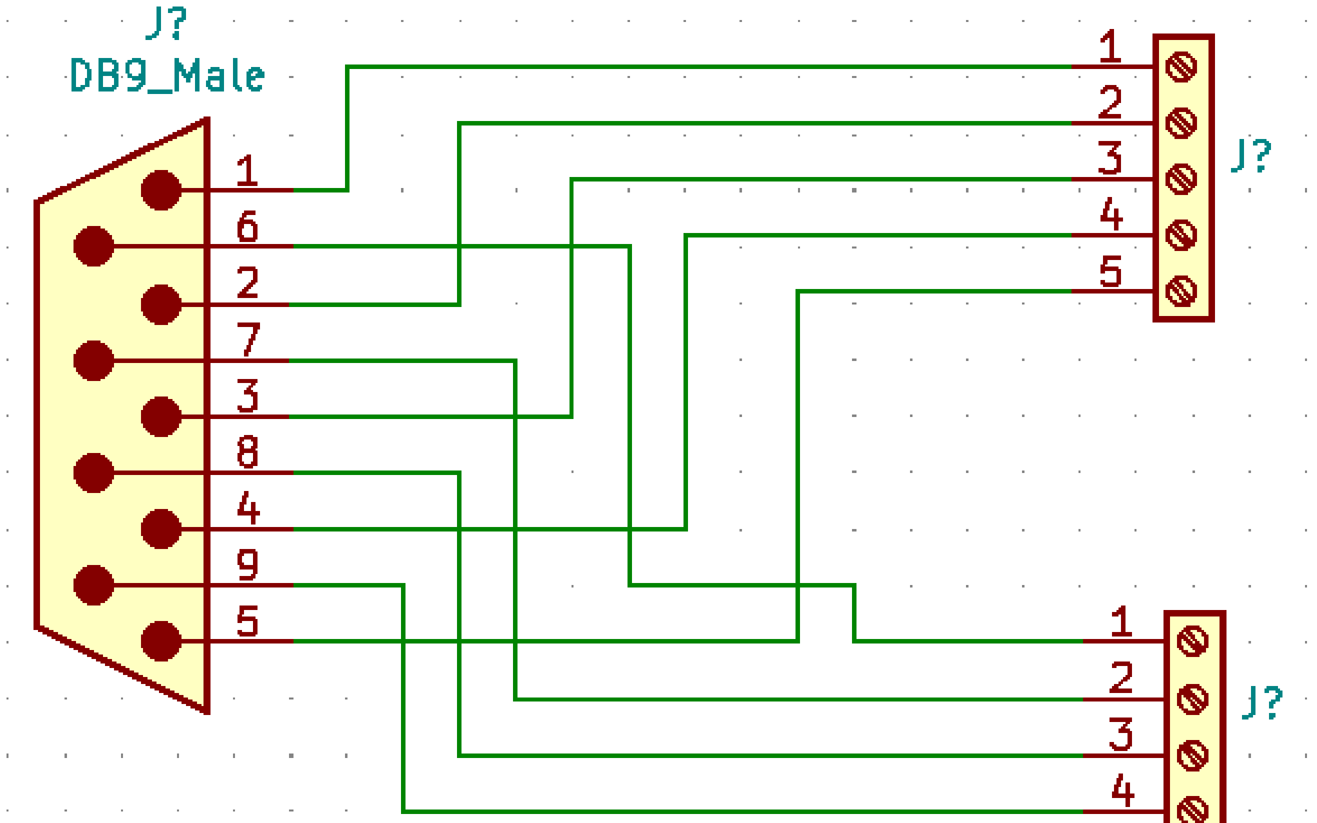
Zoom In for Detail
on “Bubbles”



Click in the Middle
of the “Bubble”

Schematic Layout Editor

Schematic Done!

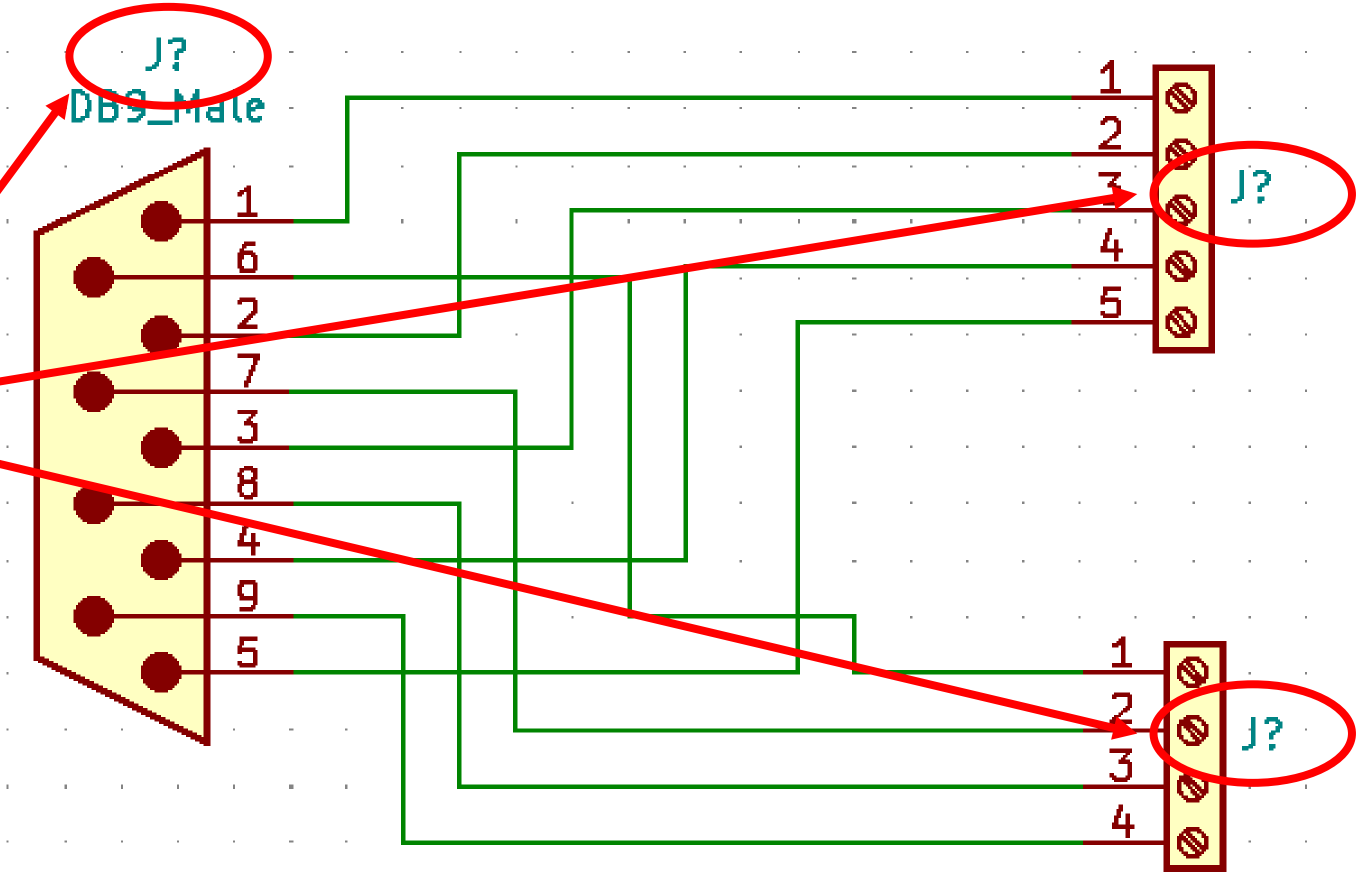


KiCad – Work Flow

- 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
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

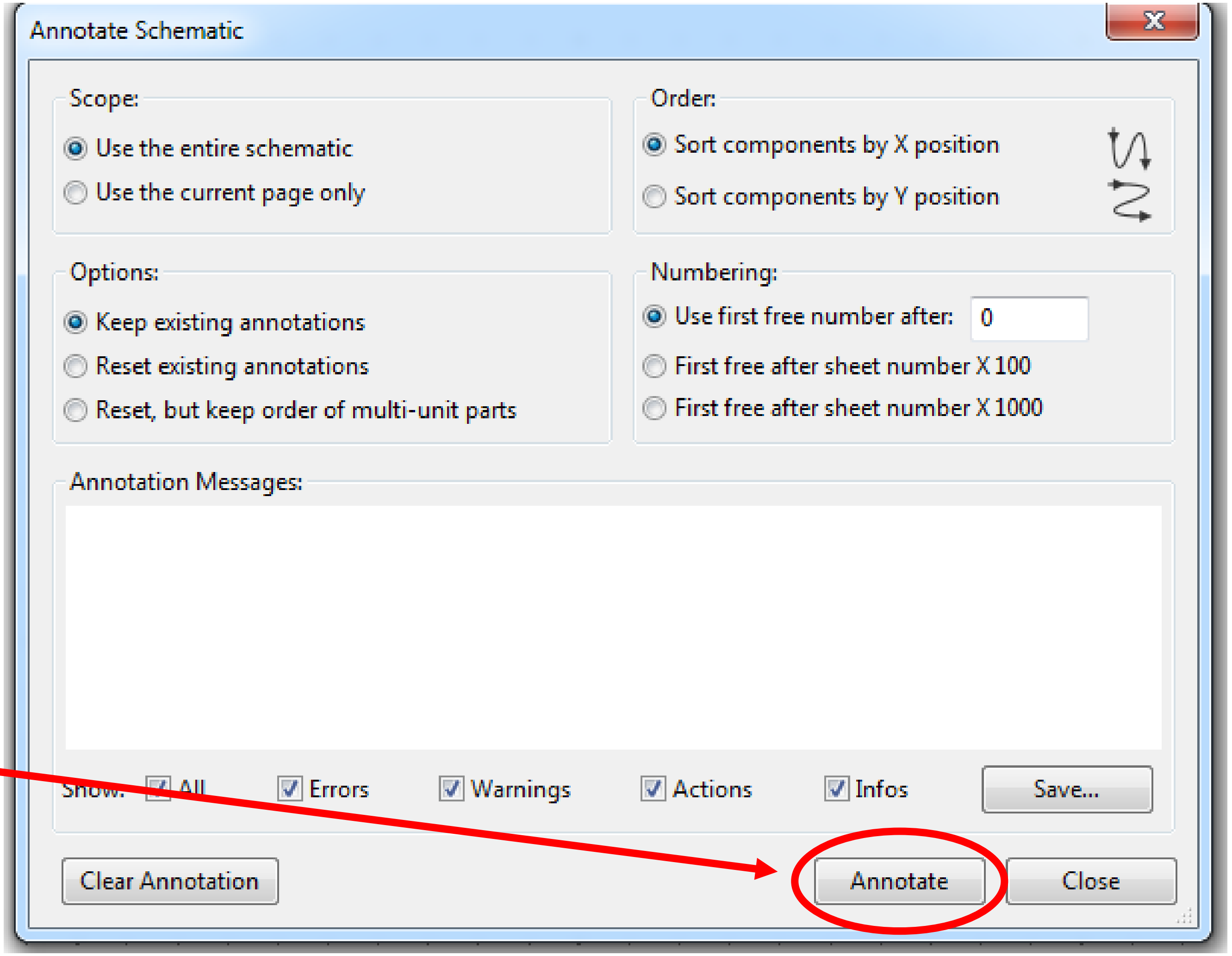
Need to Annotate



Schematic Layout Editor

Click Annotate Icon

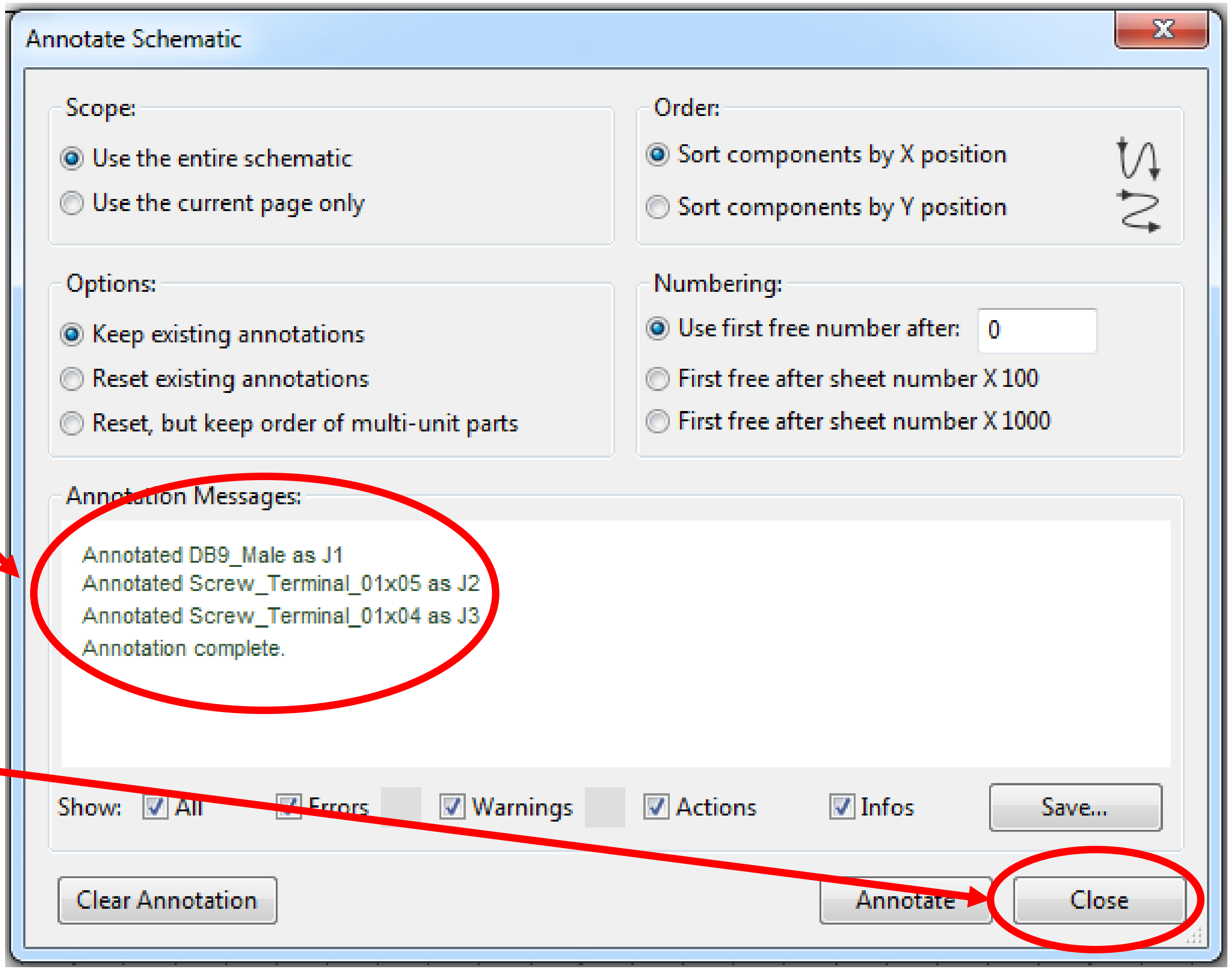
Click Annotate on the dialog



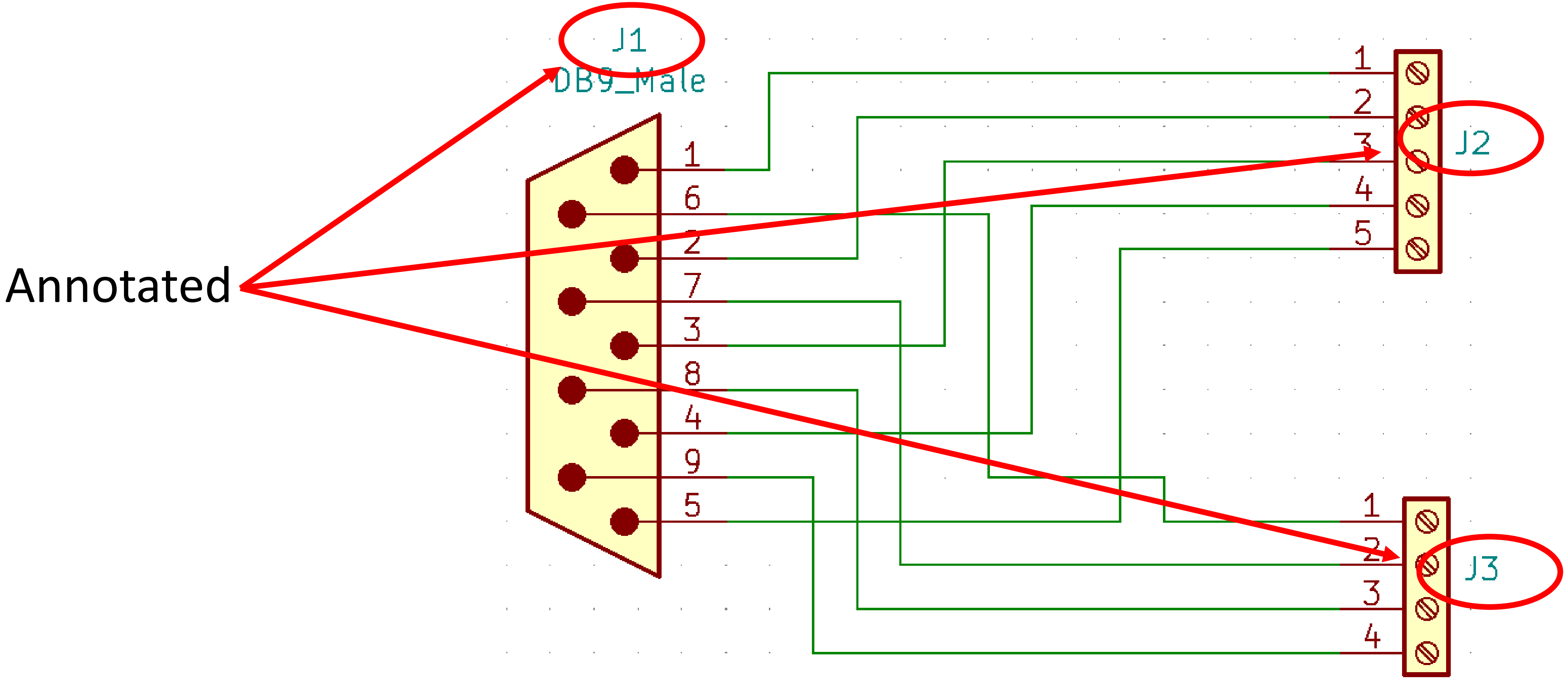
Schematic Layout Editor

Annotation Results

Click Close



Schematic Layout Editor

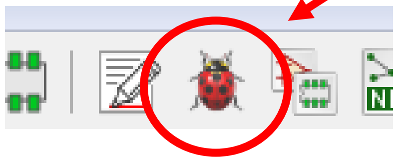


KiCad – Work Flow

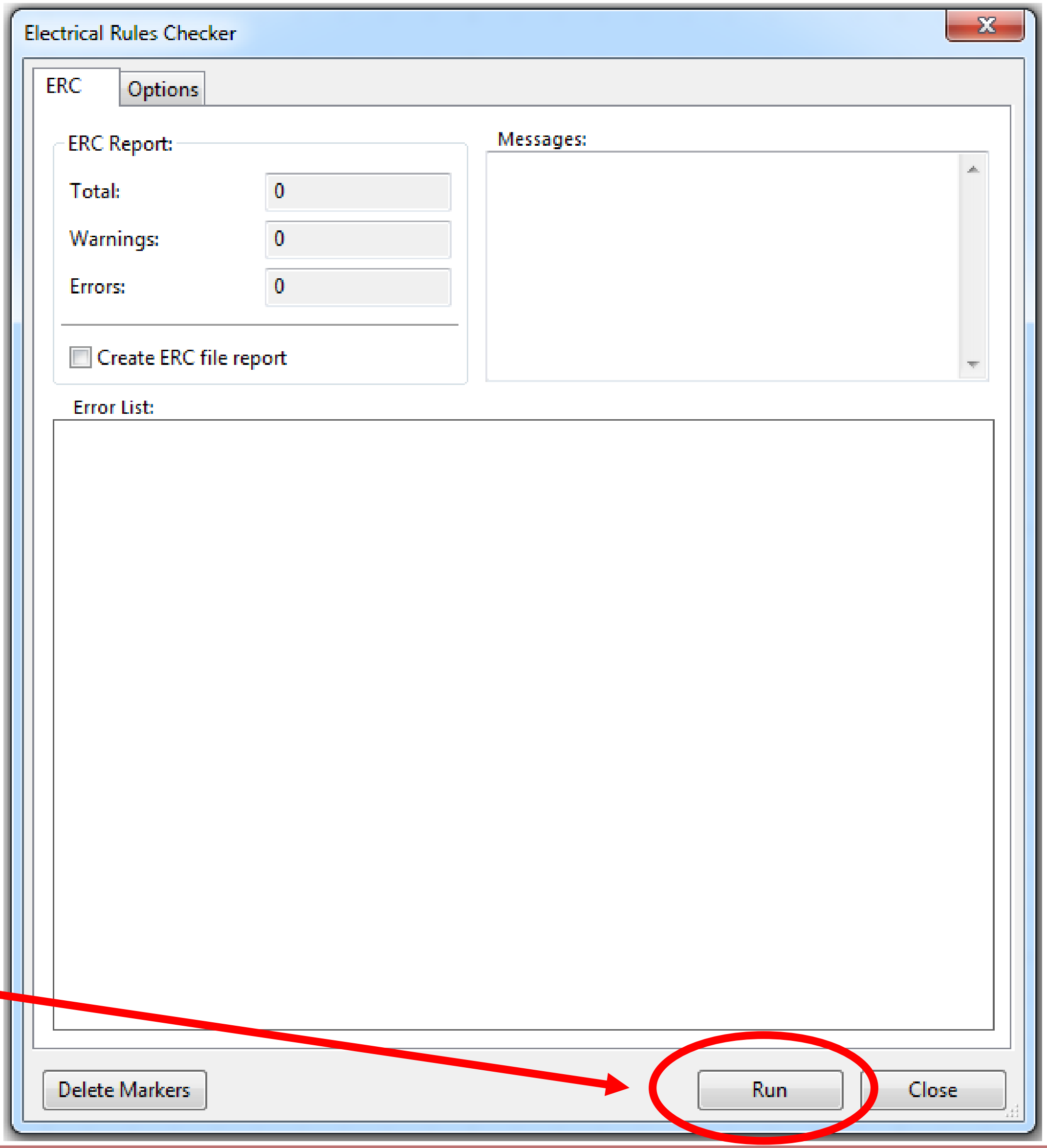
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate Footprint With Component
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

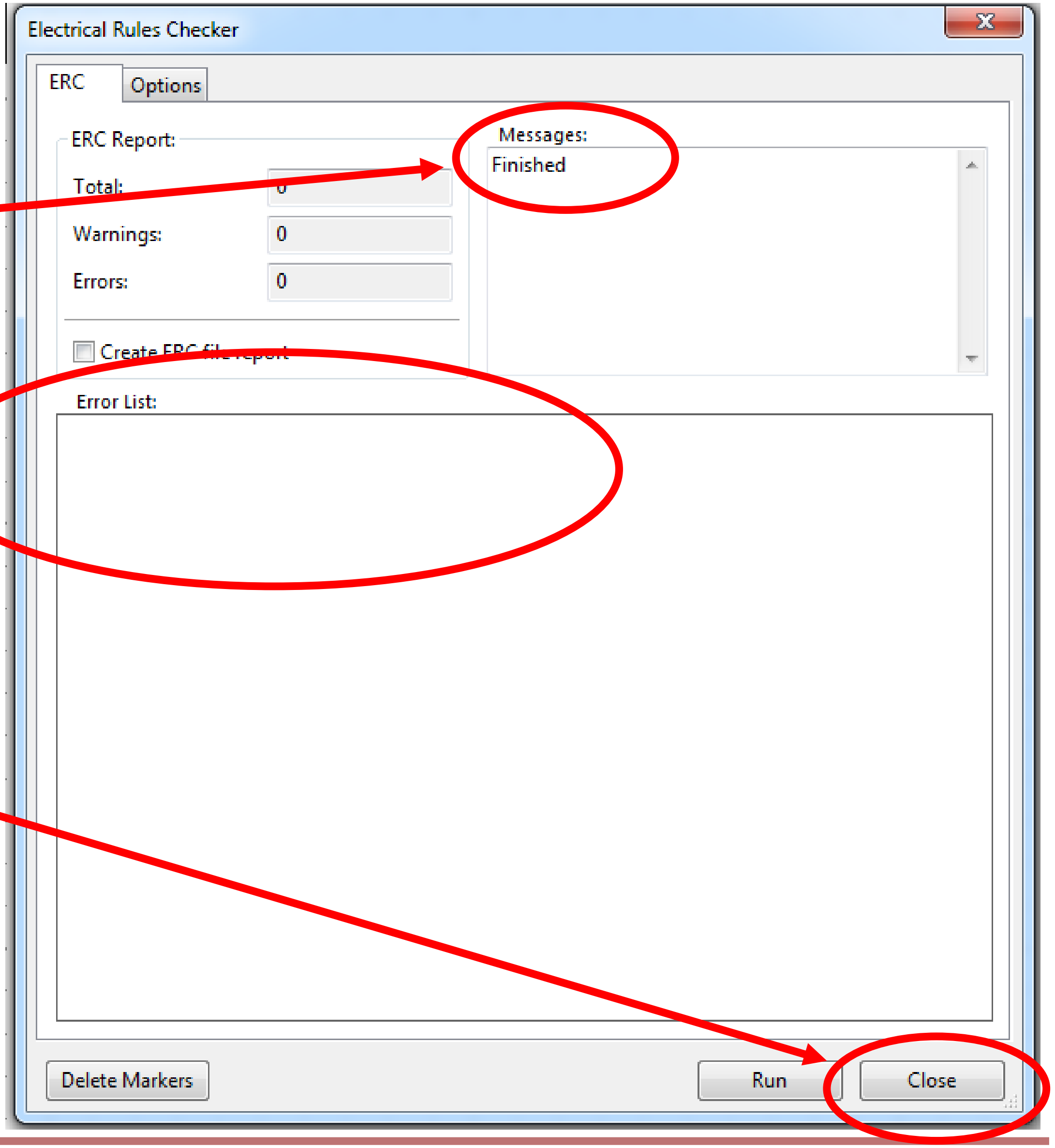


Schematic Layout Editor

Finished, No Errors

Check for Errors

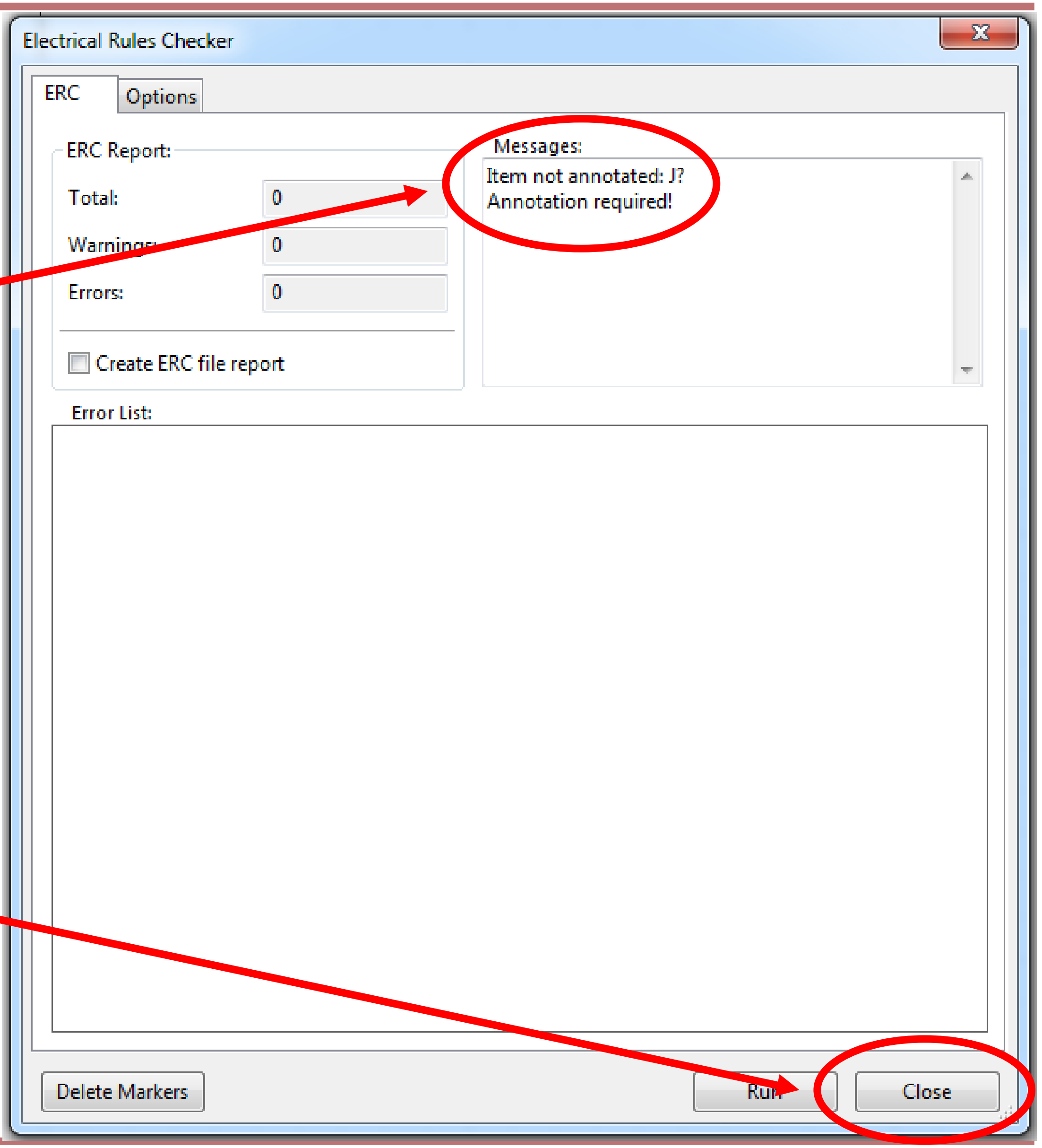
Click Close



Schematic Layout Editor

Example of Running the Electrical Rule Check Before Annotation

Click Close

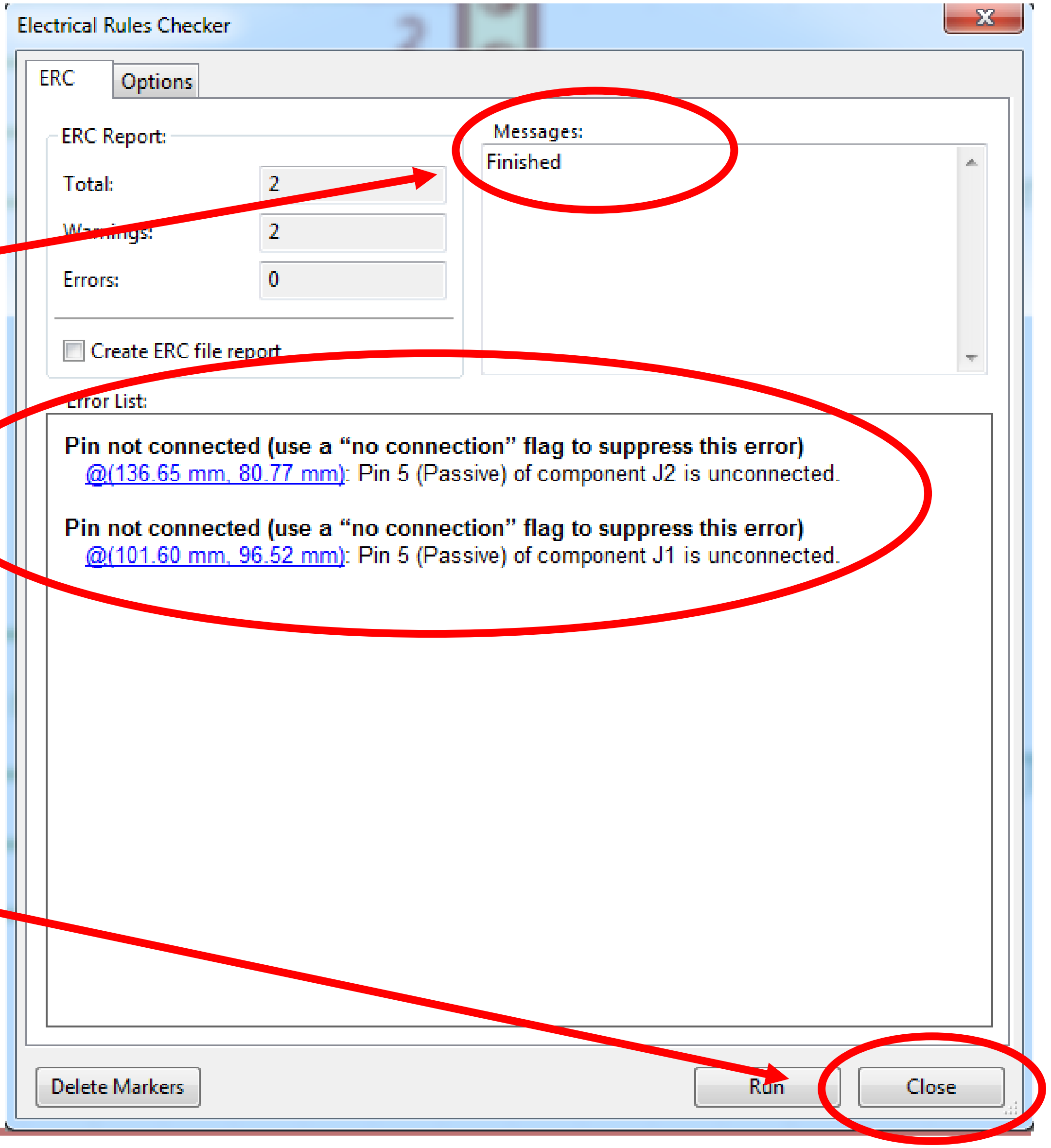


Schematic Layout Editor

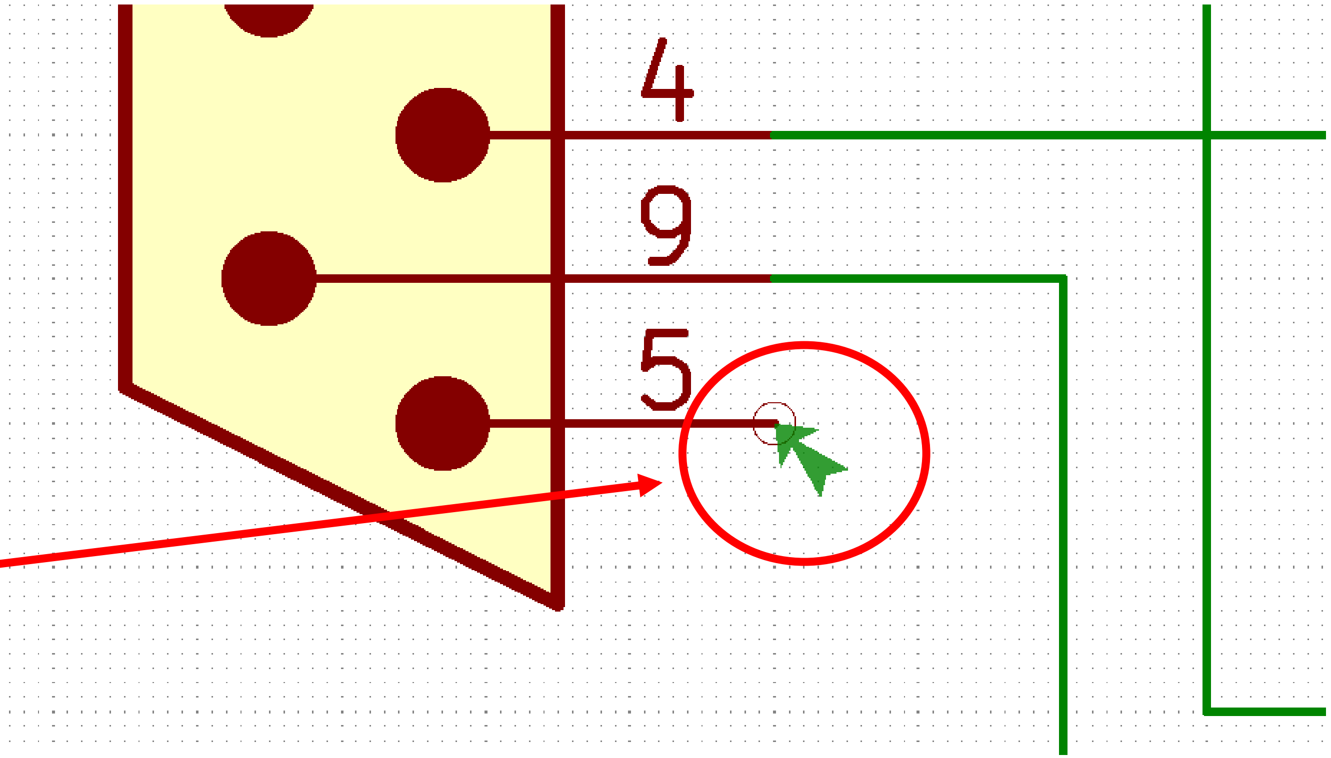
Example of Running the Electrical Rule Check with a Missing Connection

Check for Errors

Click Close



Schematic Layout Editor



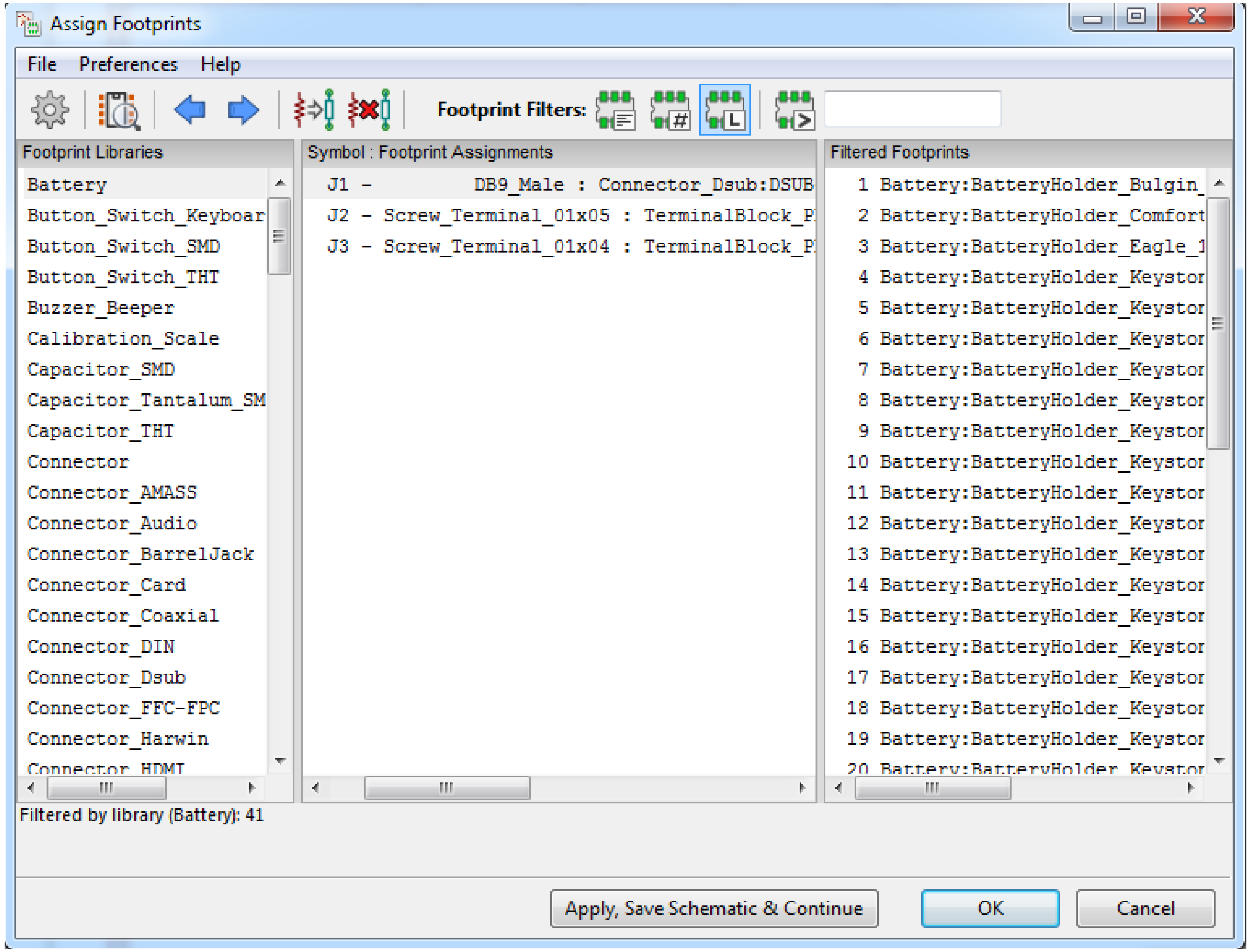
Green Marker
Arrows Point to Rule
Check Issues

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
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

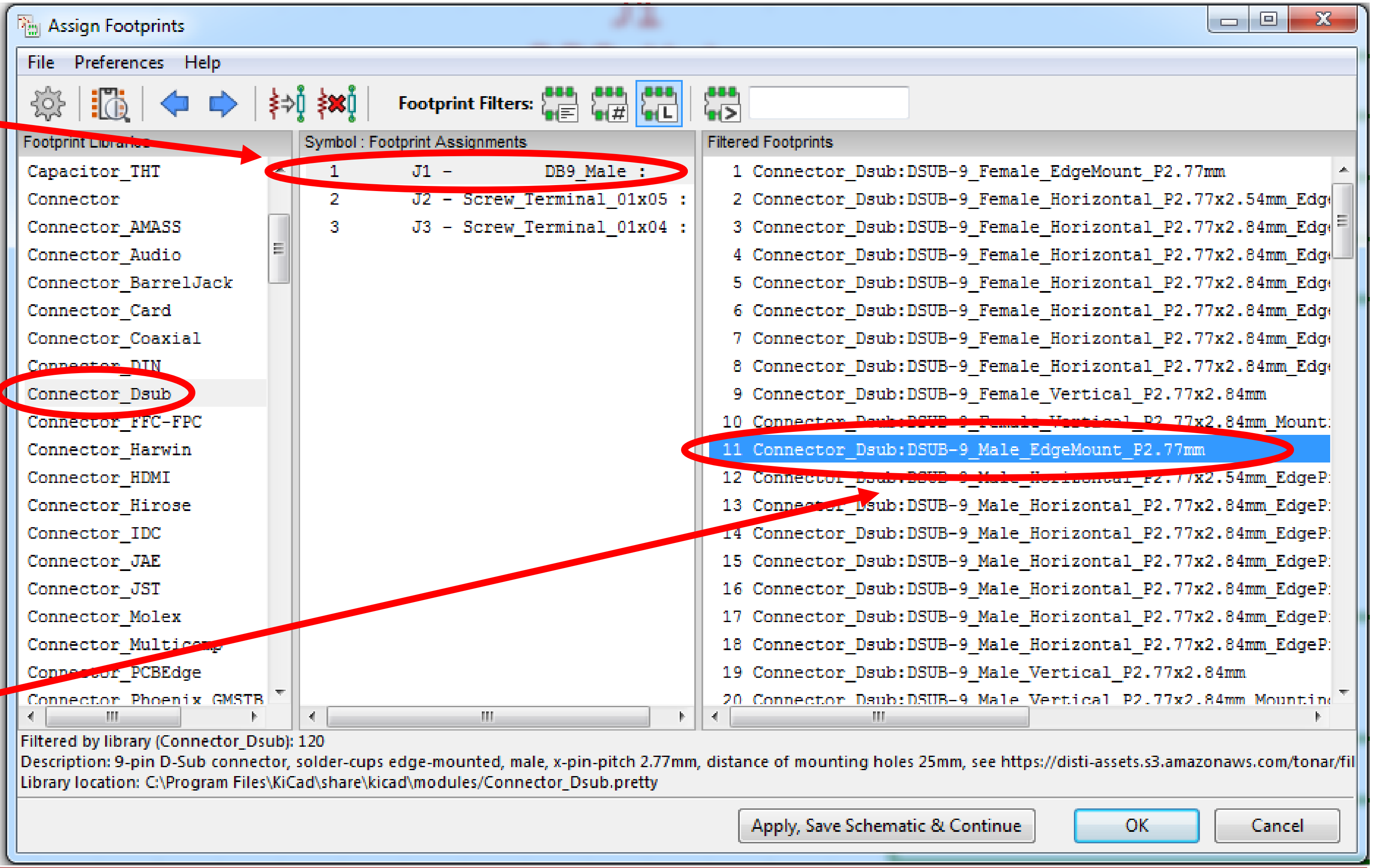


Schematic Layout Editor

Click J1

Select Library Category

Double-Click Footprint

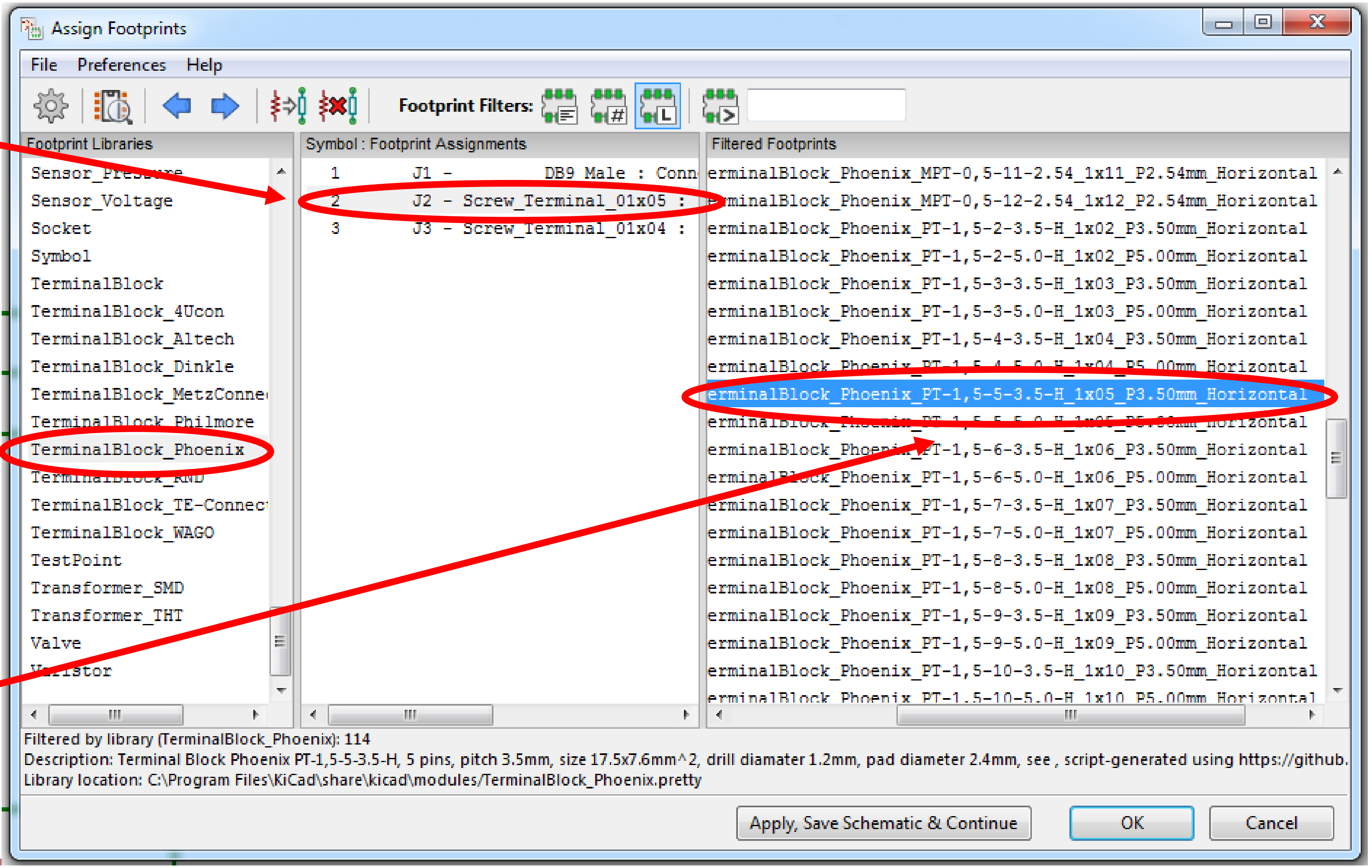


Schematic Layout Editor

Click J2

Select Library Category

Double-Click Footprint



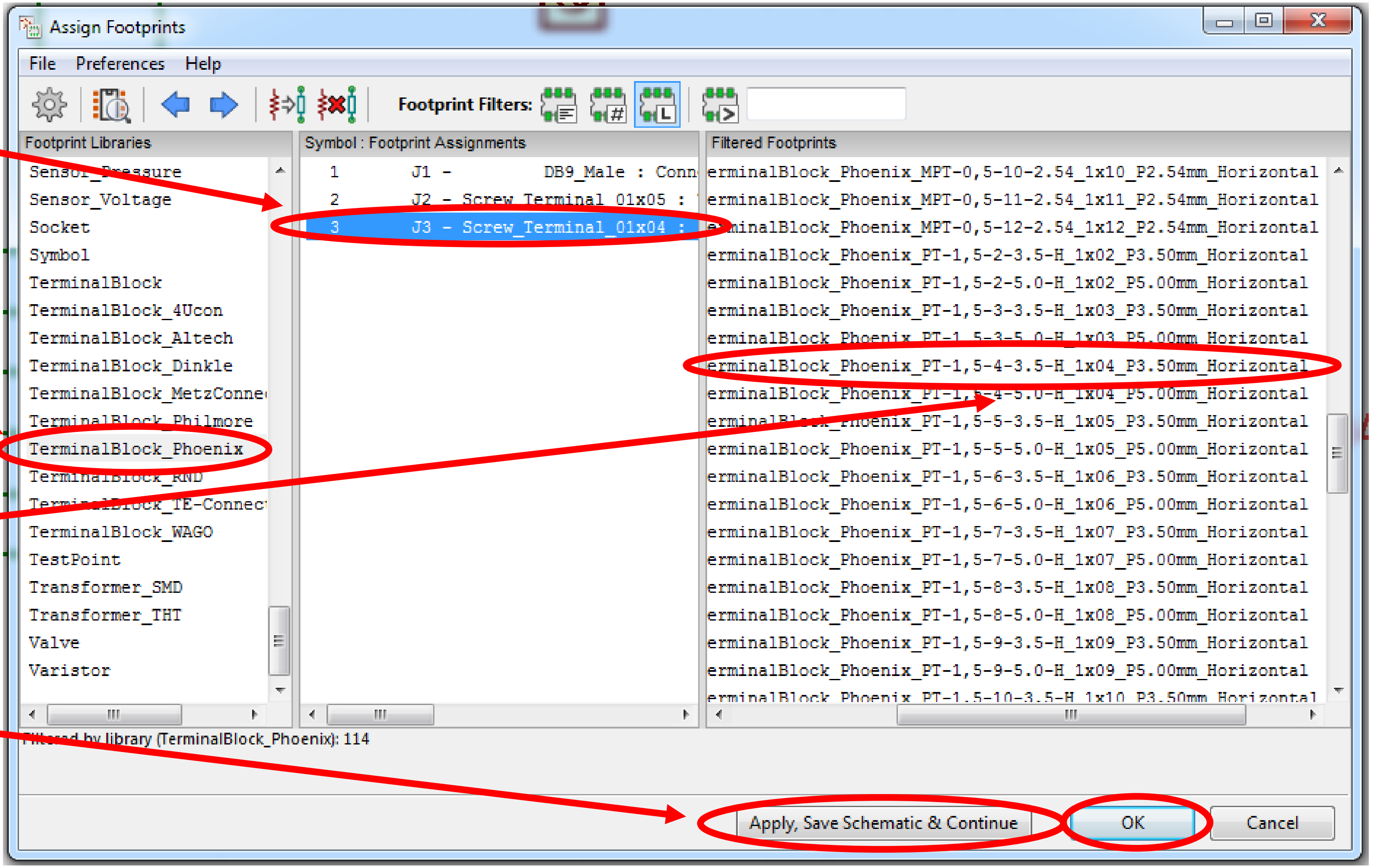
Schematic Layout Editor

Click J3

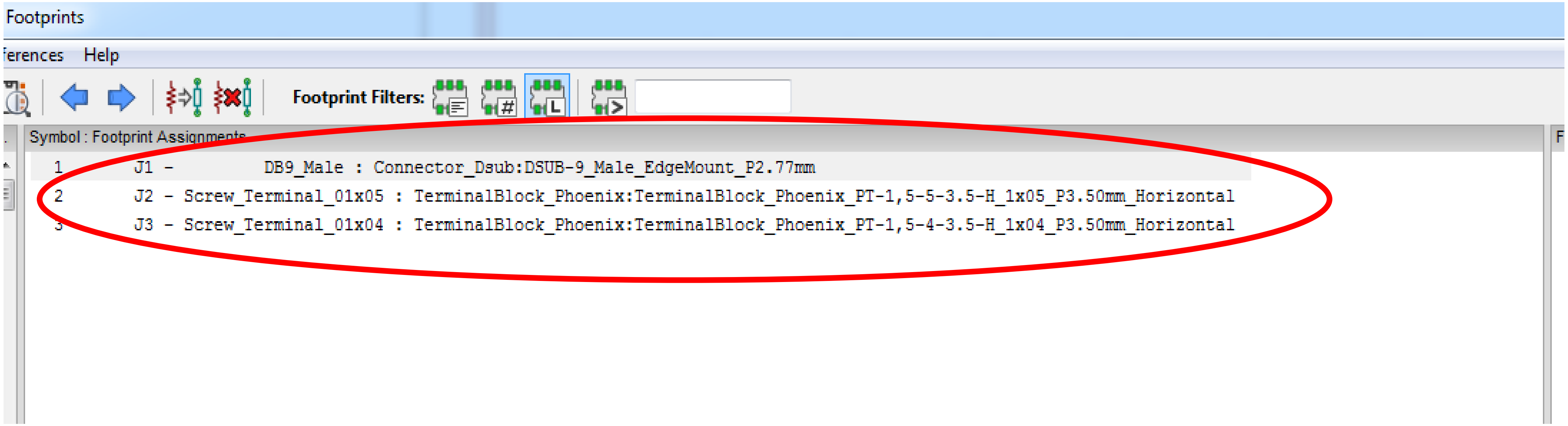
Select Library Category

Double-Click Footprint

Apply then OK



Schematic Layout Editor



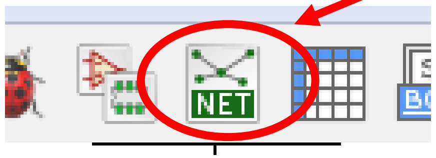
KiCad – Work Flow

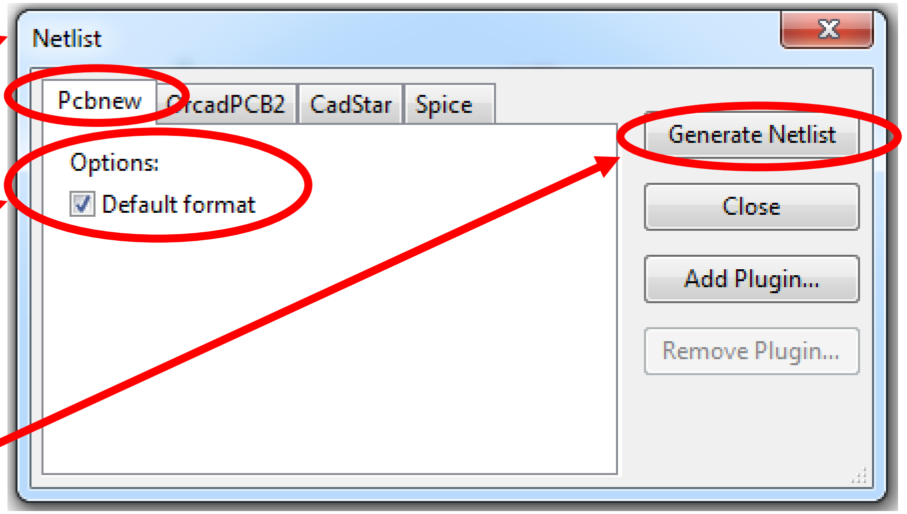
- Hand-Sketched Schematic
    - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
    - Set Page Size
    - Place Symbols from Library
        - Optional Symbol Editor
    - Annotate Schematics
    - Electrical Rule Check
    - Associate Footprint With Component
    - Generate Netlist

- PCB Layout Editor
    - Import Netlist
    - Mounting Holes
        - 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 Generate Netlist Icon





Review Dialog

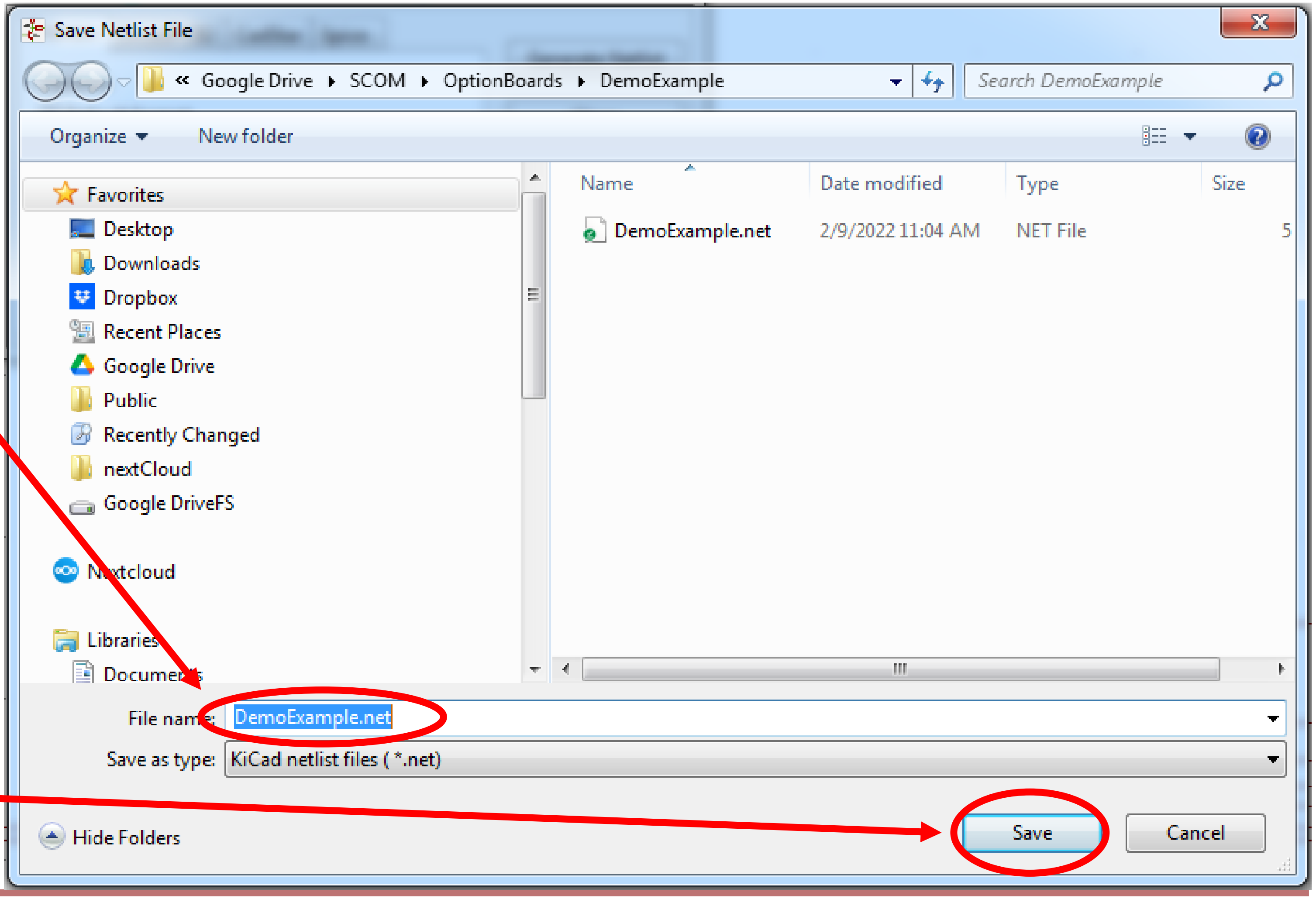
Use Defaults

Click Generate Netlist

Schematic Layout Editor

Use Default

Click Save

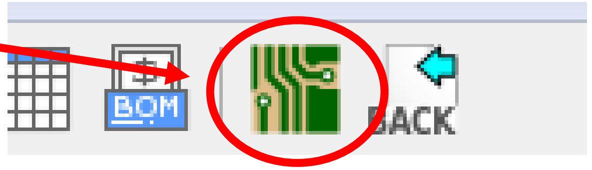


KiCad – Work Flow

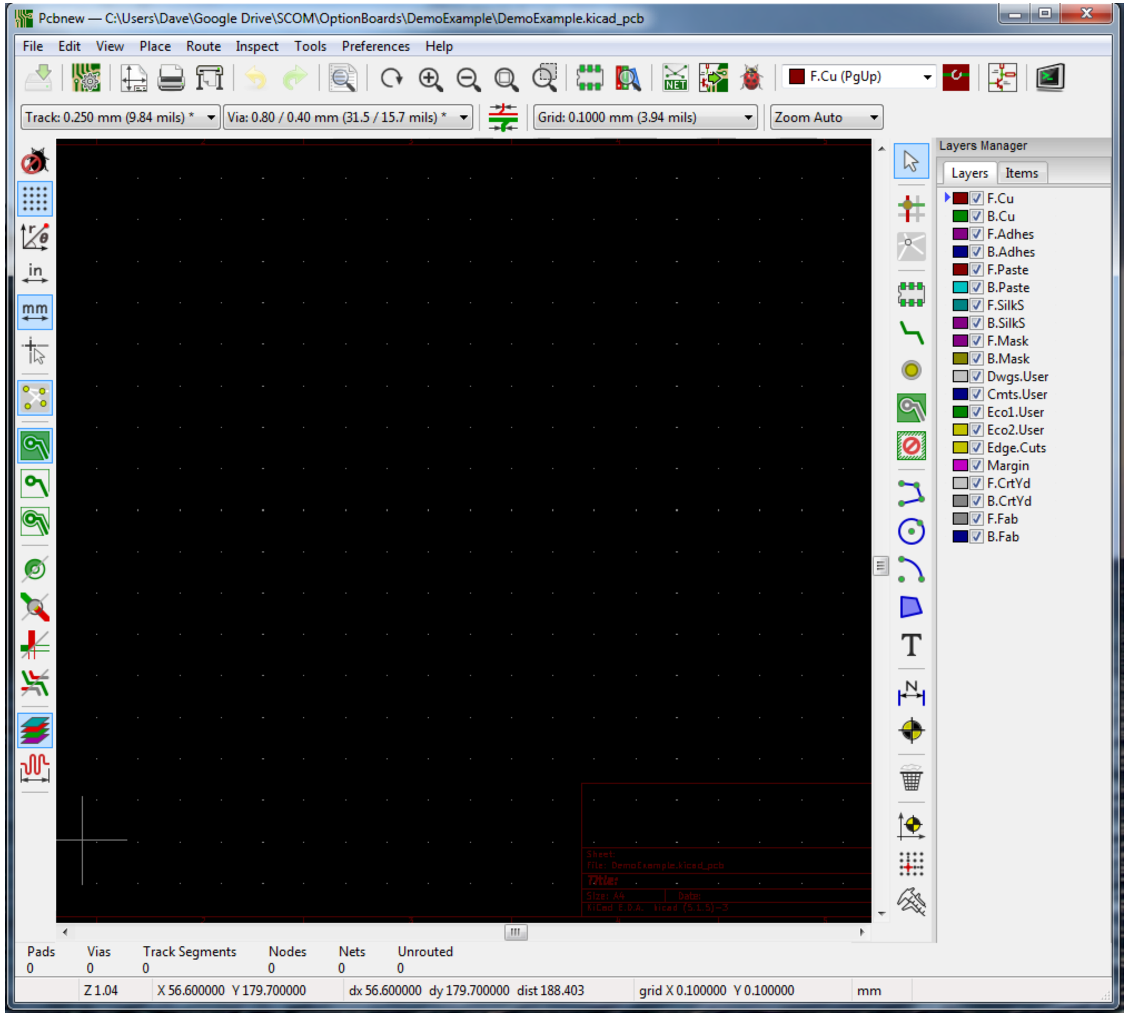
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate Footprint With Component
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

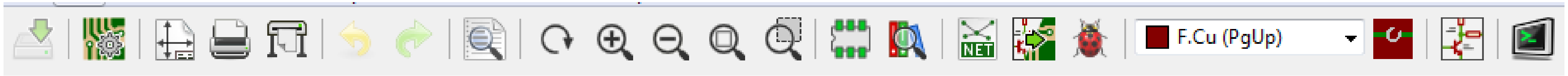


PCB Layout Editor



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

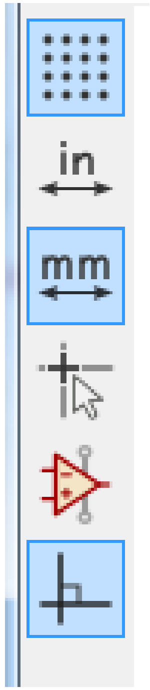
PCB Layout Editor



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

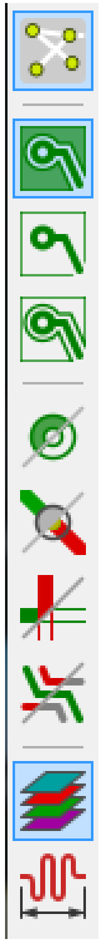
PCB Layout Editor

- Left Toolbar – Select Mode



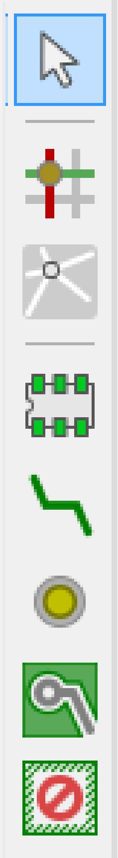
  - Turn the Grid On and Off – Blue is On
  - Switch Units to Inches – Blue is Selected
  - Switch Units to Millimeters -- Blue is Selected
  - Choose Cursor Shape – Blue is Full Screen
  - Toggle Visibility of Invisible Pins On Symbols – Blue is On
  - Toggle Free Angle vs. 90 Degrees

PCB Layout Editor



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

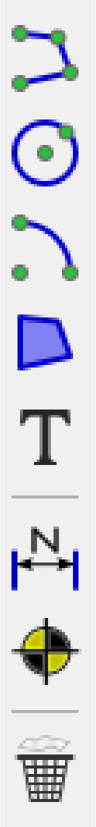
PCB Layout Editor



Right Toolbar

- Select Item
- Highlight Net
- Display Local Rats Nest
- Add Footprints
- Route Tracks
- Add Vias
- Add Filled Zones
- Add Keepout Areas

PCB Layout Editor

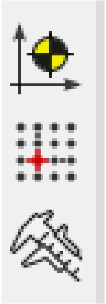


Right Toolbar

- Add Graphic Lines
- Add Graphic Circles
- Add Graphic Arcs
- Add Graphic Polygons
- Add Text
- Add Dimension
- Add Layer Alignment Target
- Delete Item

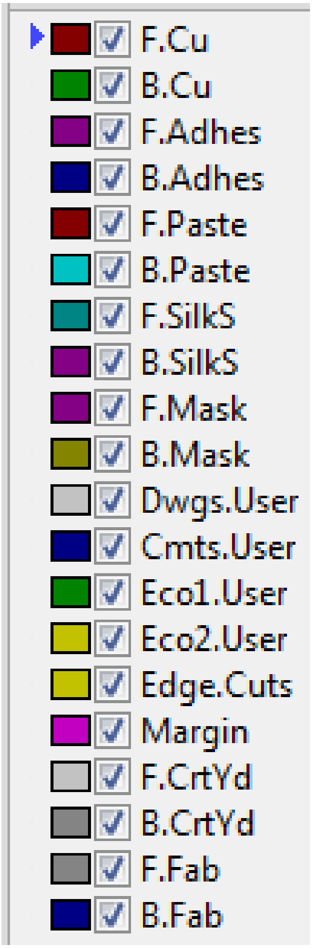
PCB Layout Editor

Right Toolbar



- Place Axis Origin (Only for some plot formats)
- Set Grid Origin
- Measure Distance

PCB Layout Editor



Layer Manager

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

PCB Layout Editor

Auxiliary Toolbar Mode Selections

- Track Width
- Via Size
- Auto Track Width Select
- Grid Step Size
- Zoom Step Size

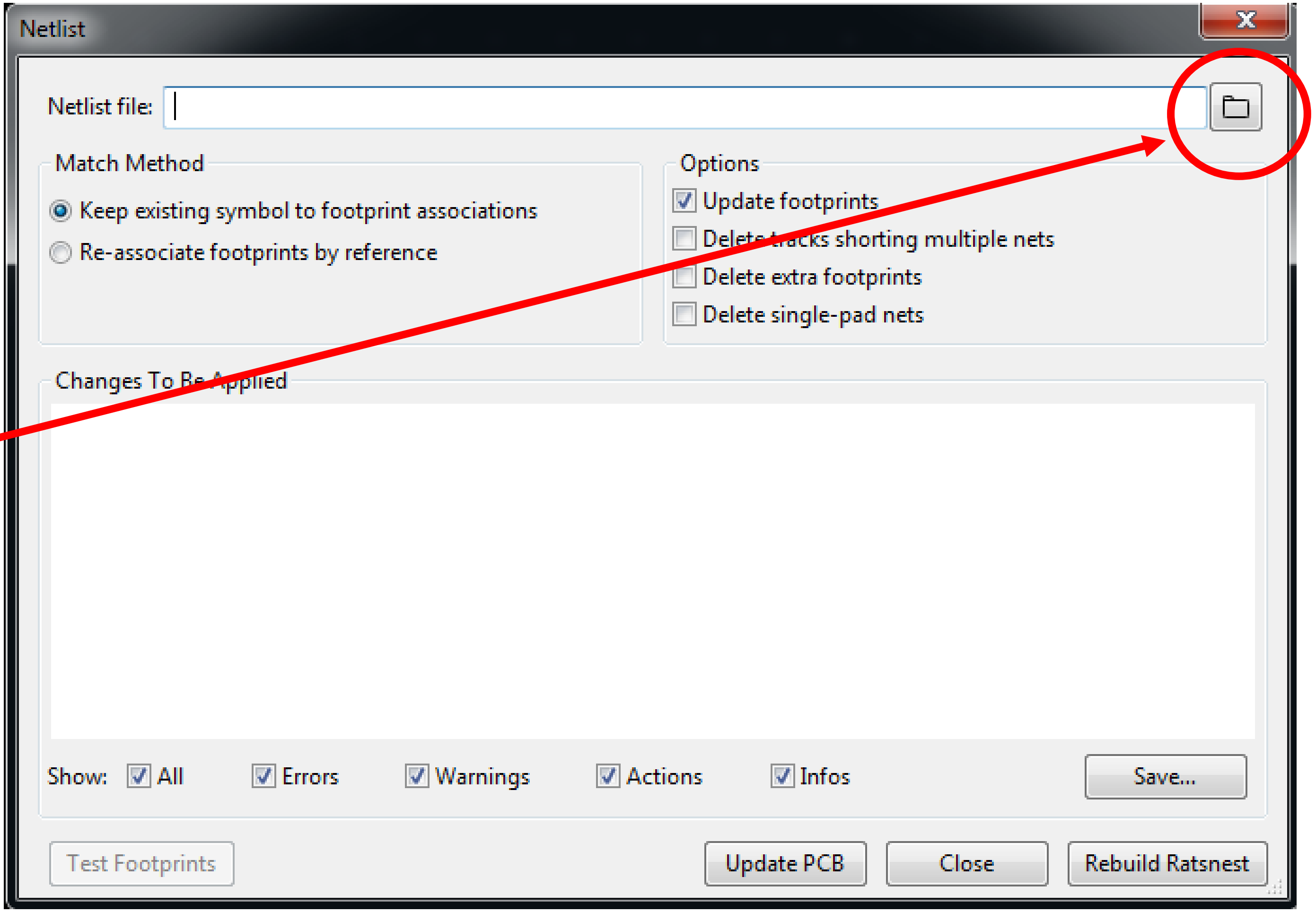
KiCad – Work Flow

- 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
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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 Load Netlist

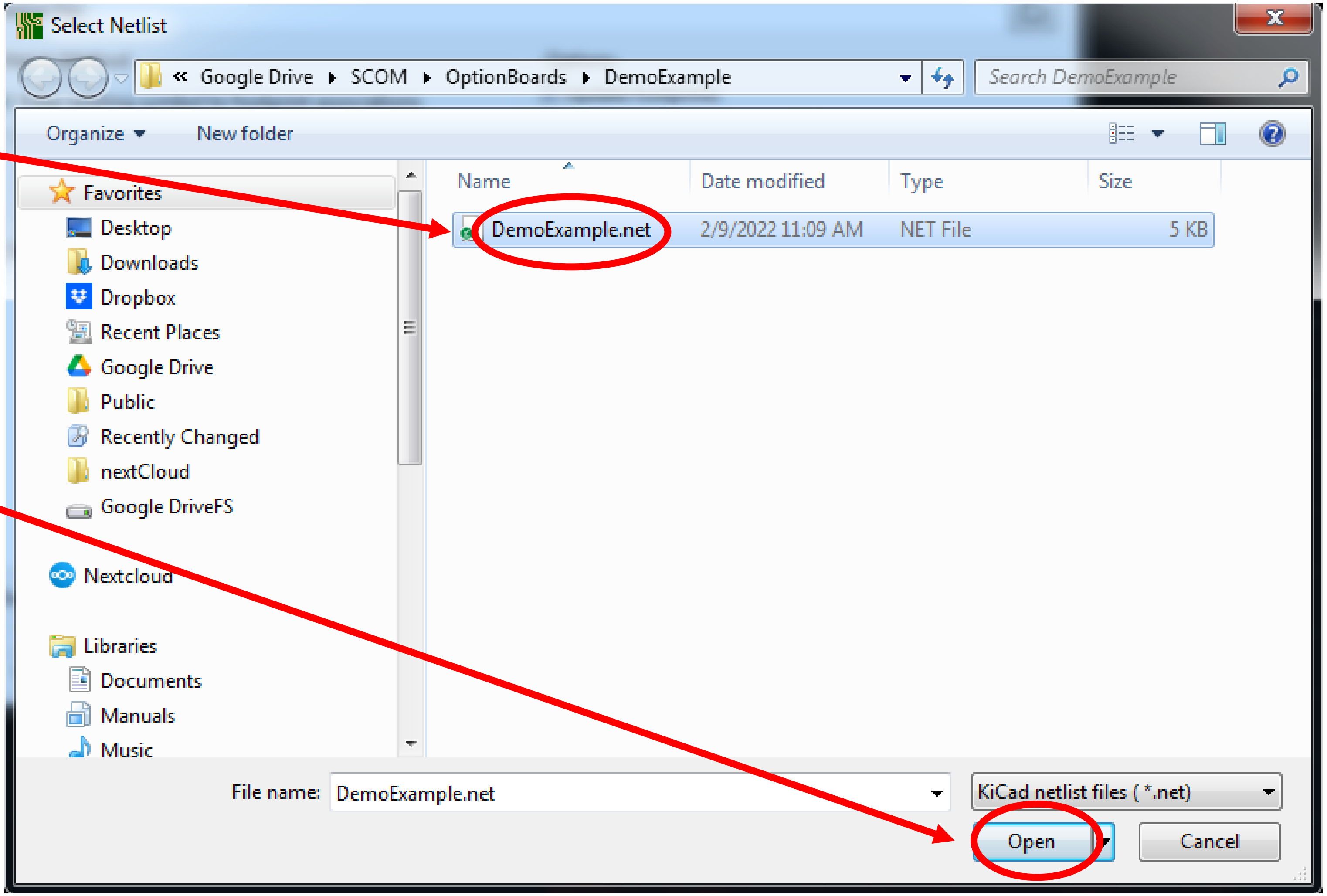
Select File



PCB Layout Editor

Select File

Click Open

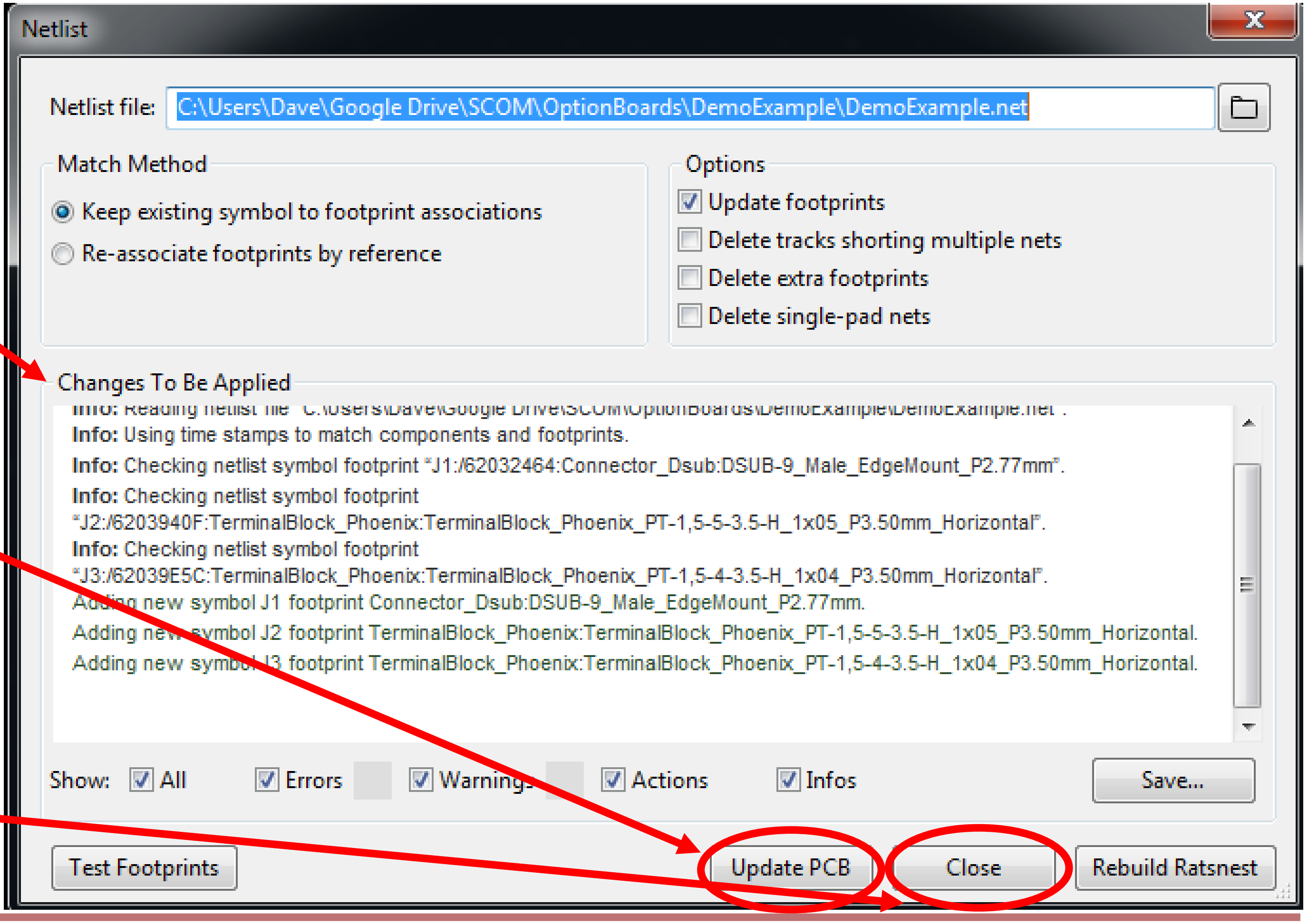


PCB Layout Editor

Load Status

Update PCB

Then Close

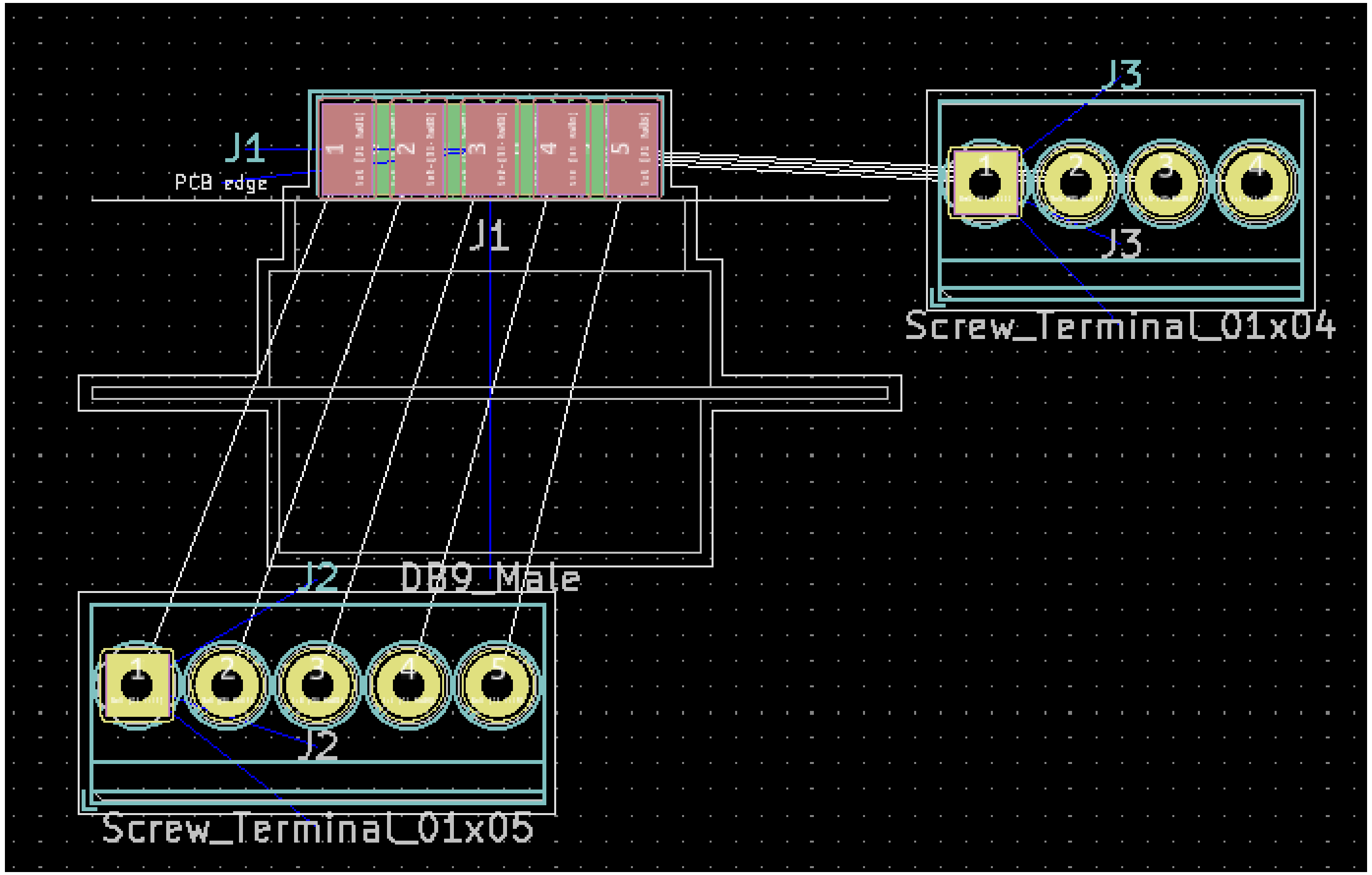


PCB Layout Editor

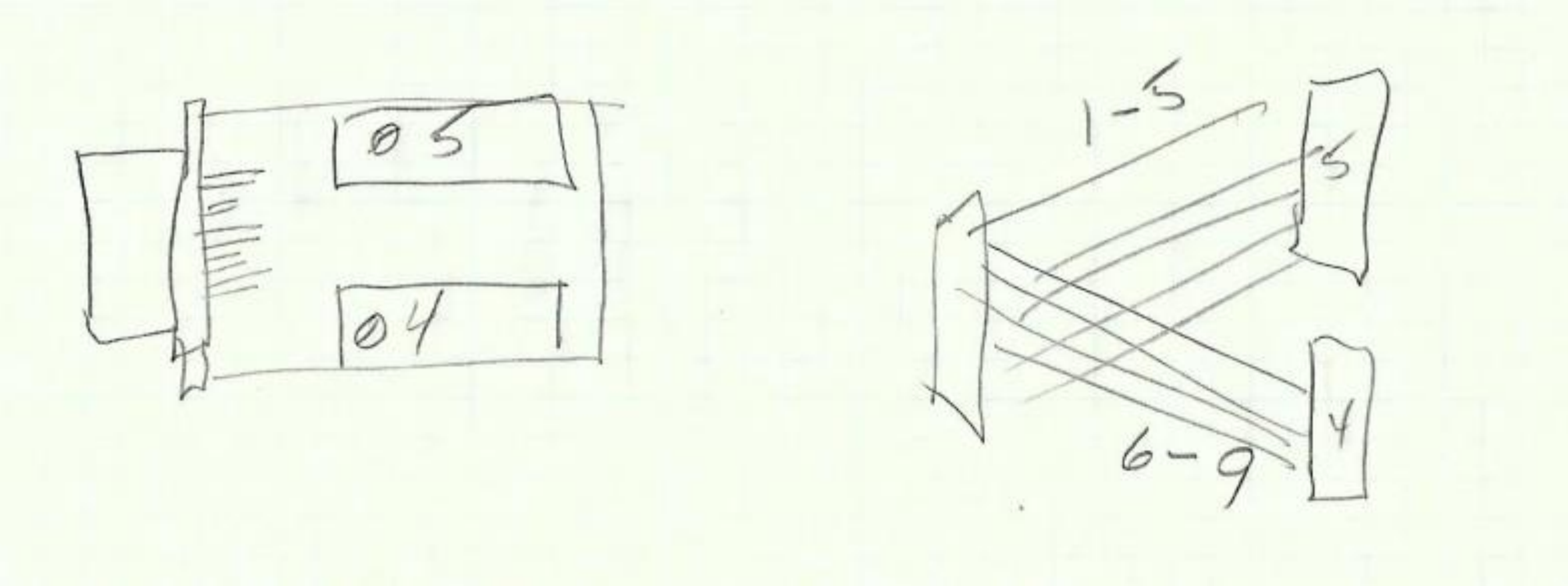
Click to place the Rats in the Drawing Area

Click, press M and Move each Component to Position

R to Rotate



Example Project – Hand Sketch

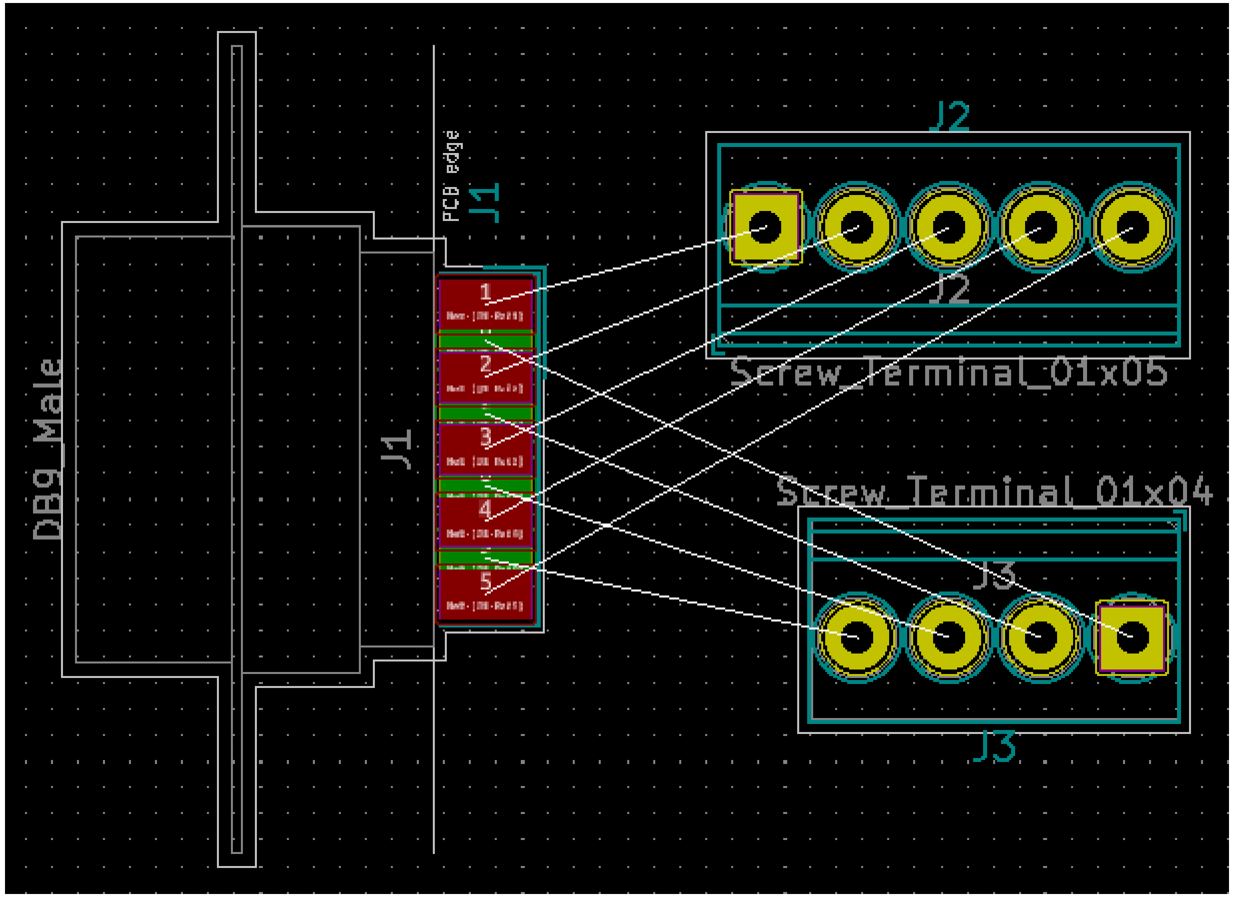


PCB Layout Editor

The View is Always from the Top of the Board

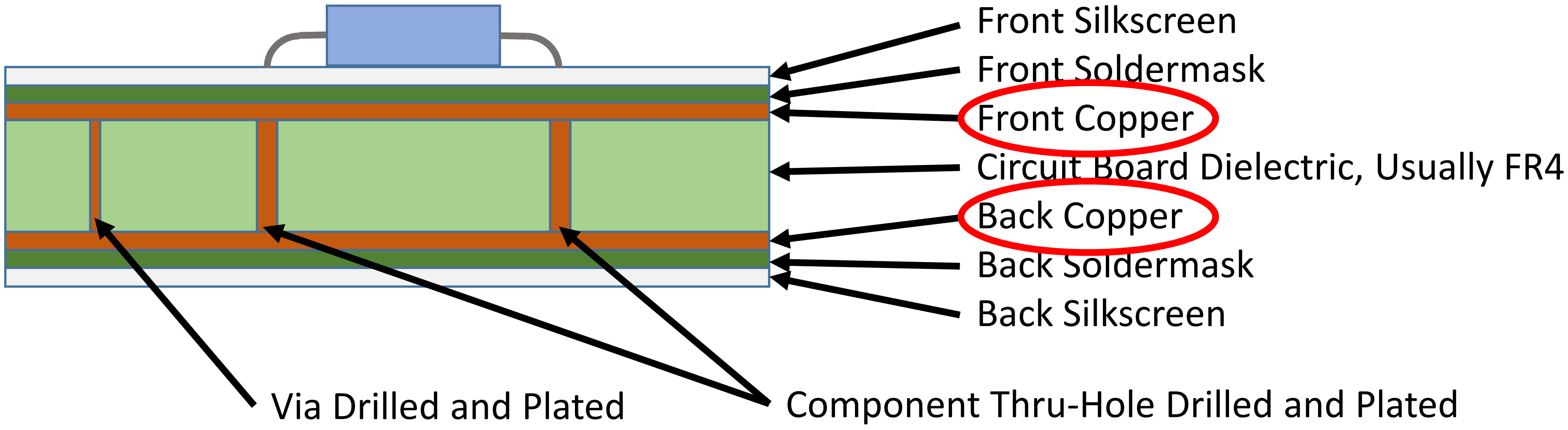
Components placed

The white lines are the Air Wires. These are the Traces to be Routed.

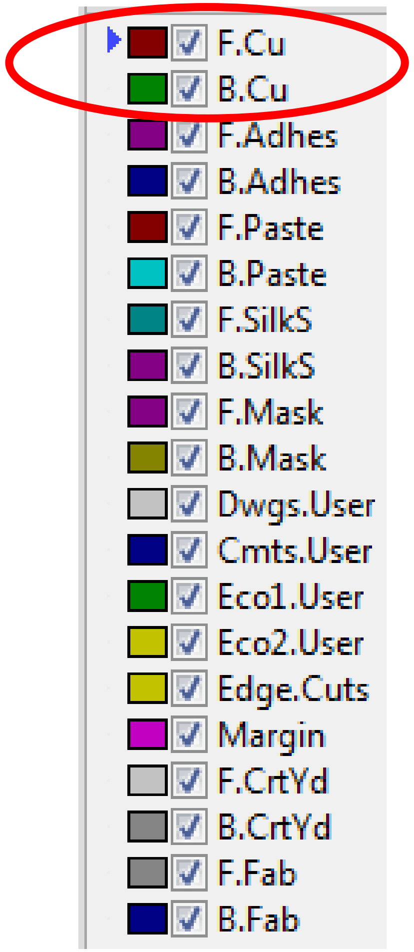


Anatomy of a PCB

- 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



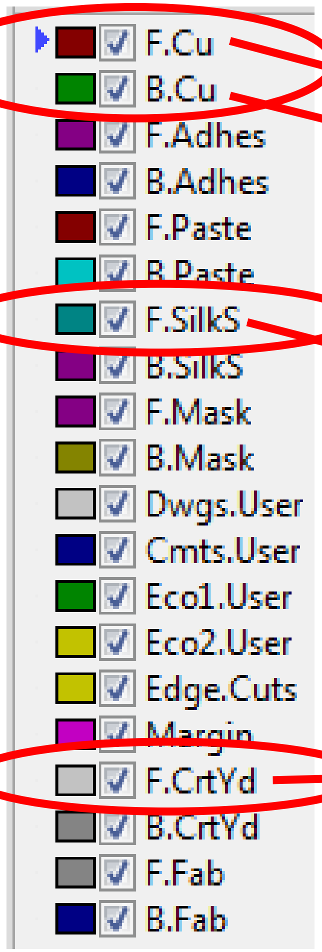
PCB Layout Editor

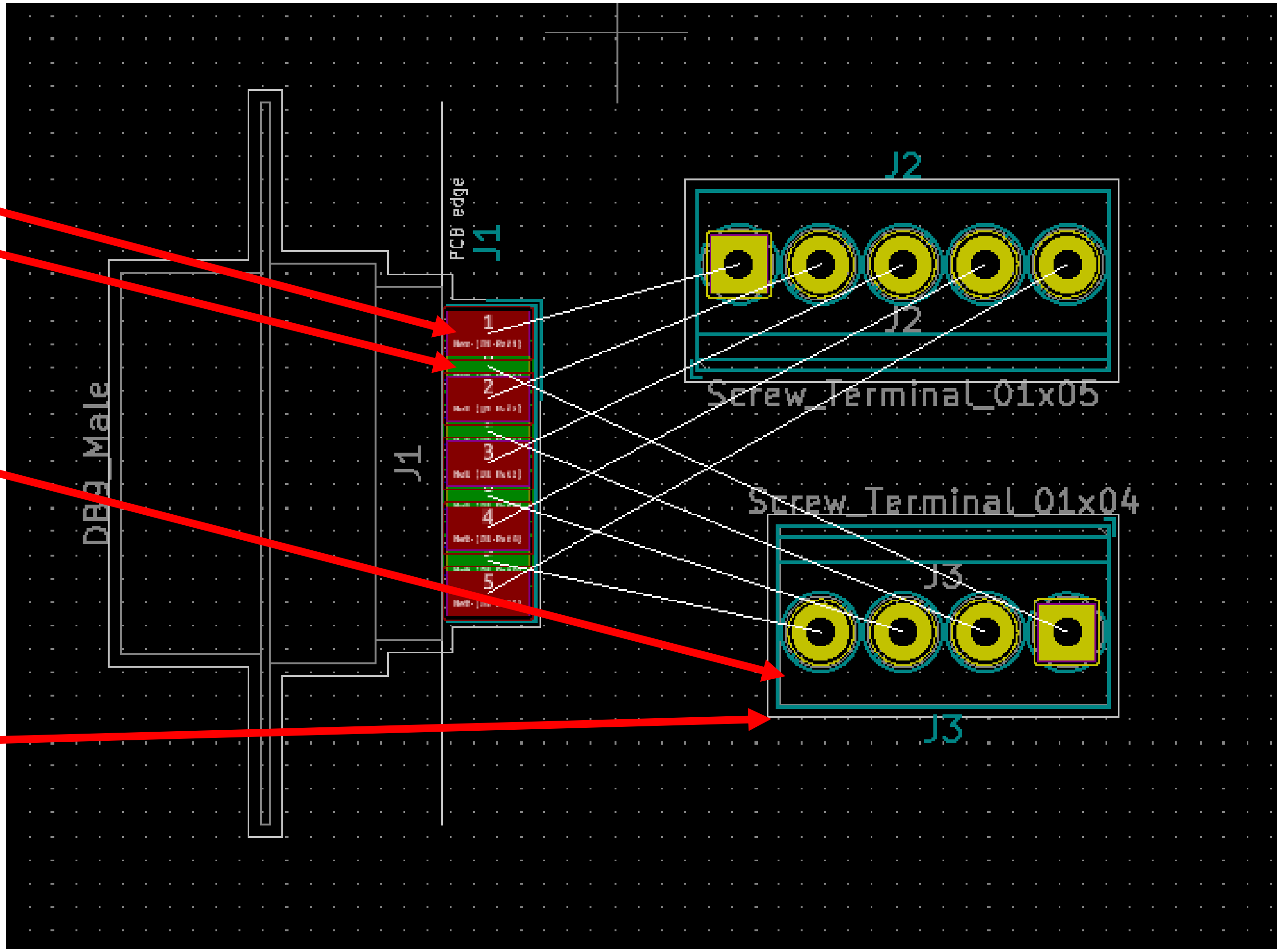


Layer Manager

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

PCB Layout Editor





KiCad – Work Flow

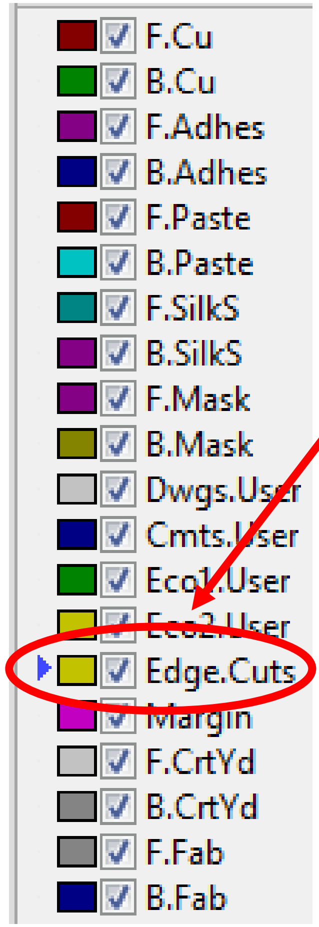
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate Footprint With Component
  - Generate Netlist

- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

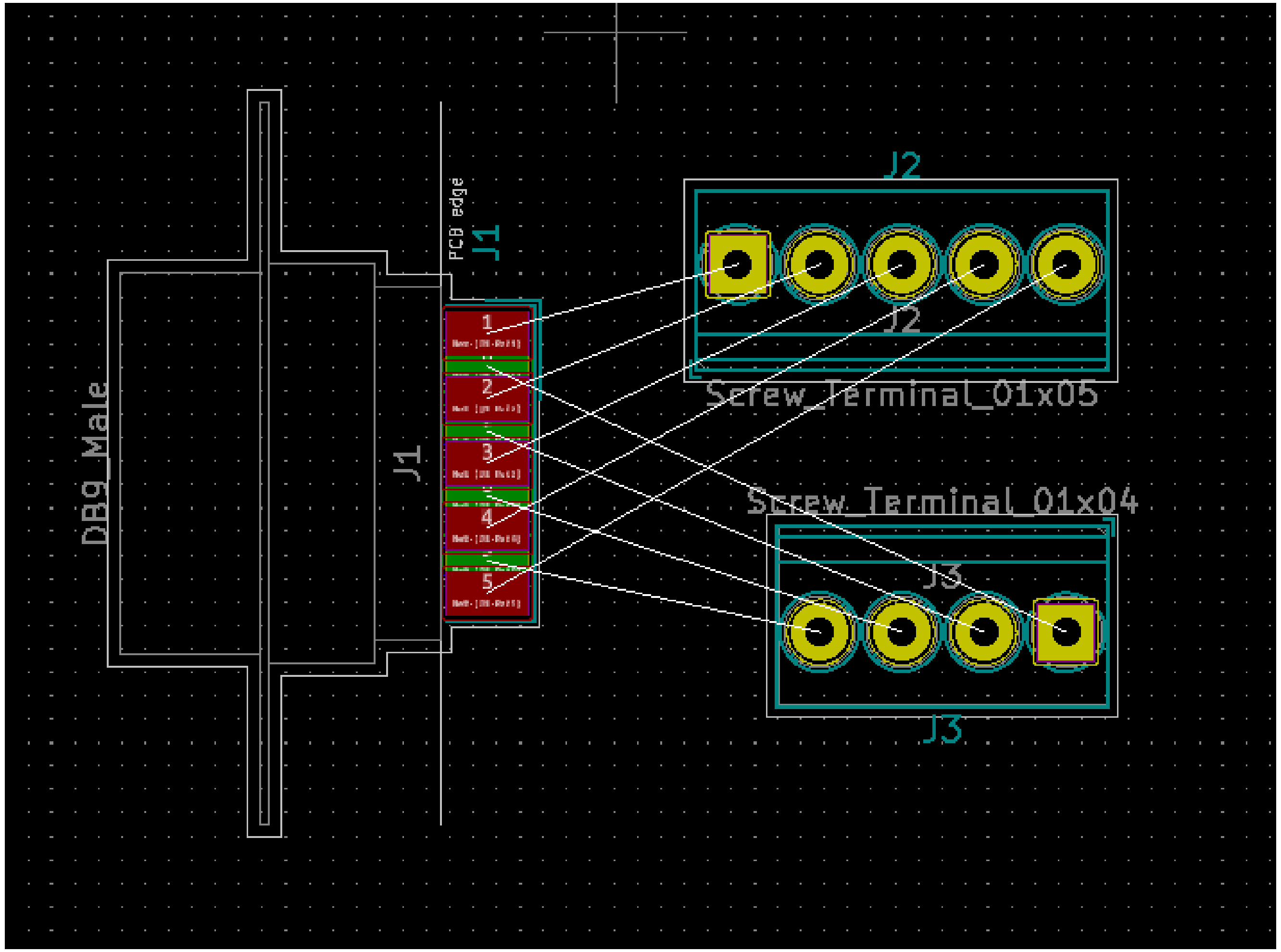
KiCad – Work Flow

- 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
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

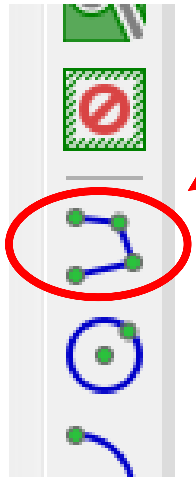


Select the Edge.Cuts Layer



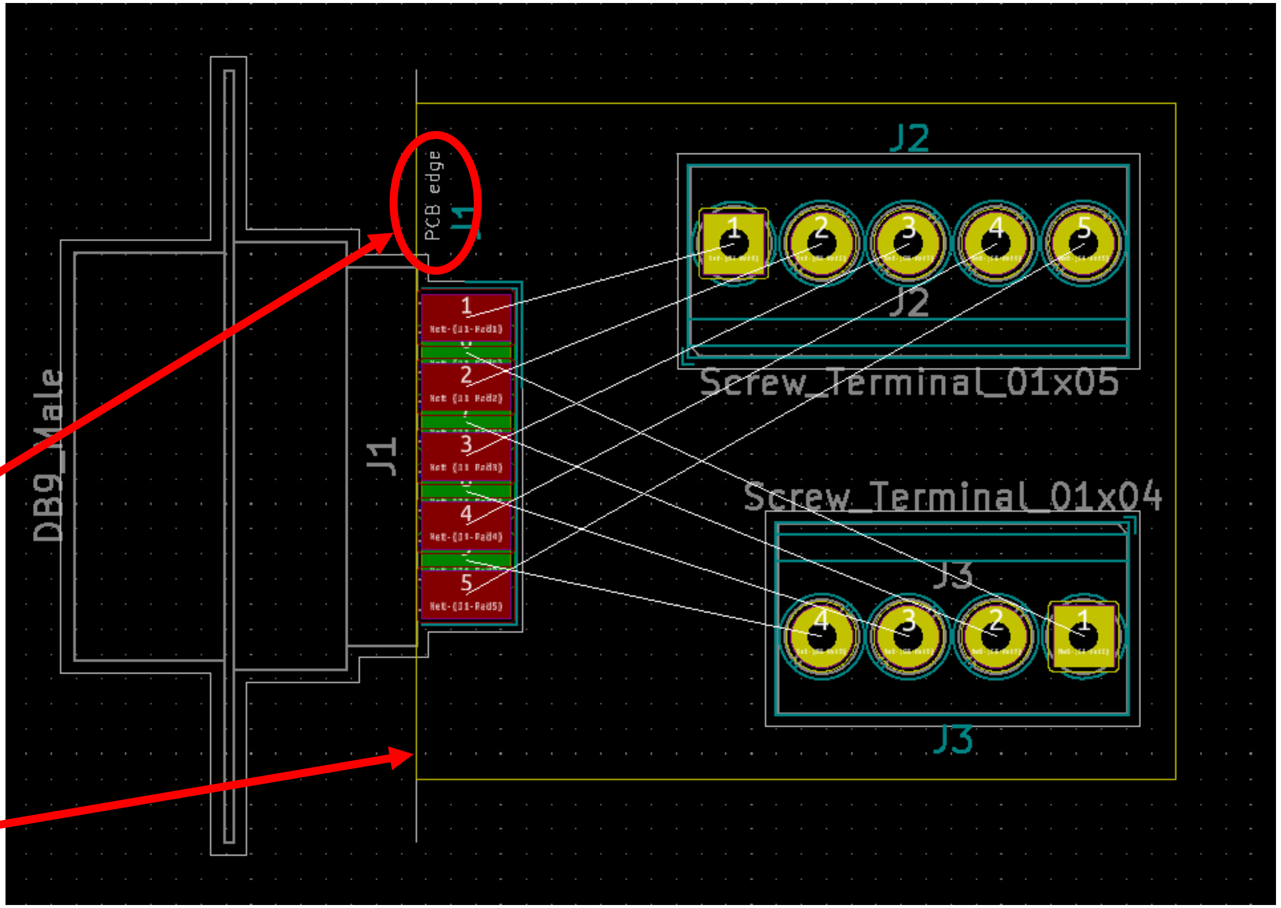
PCB Layout Editor

Click Add Graphics Lines



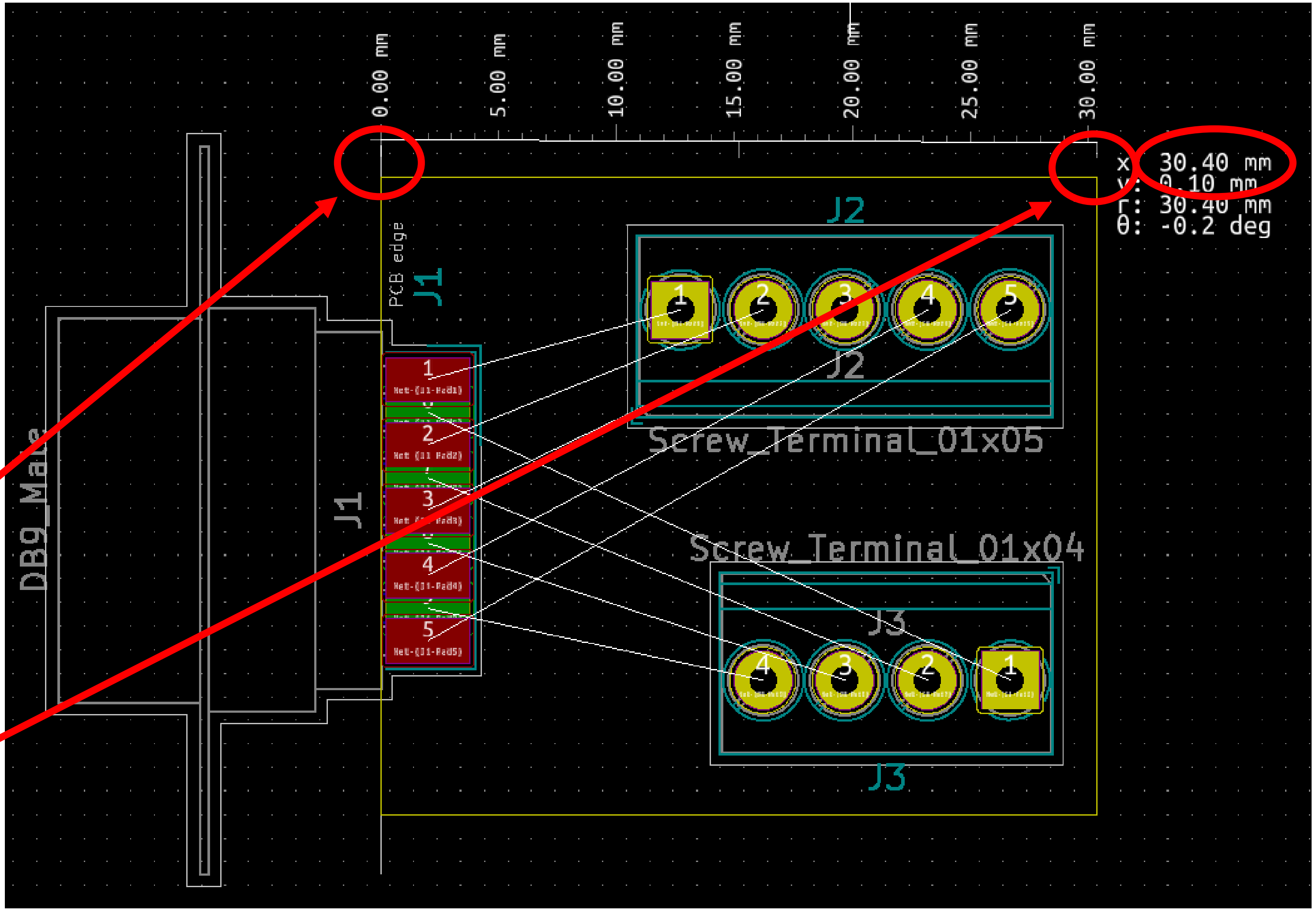
Note “PCB Edge”

Draw a Complete Box Around Components



PCB Layout Editor

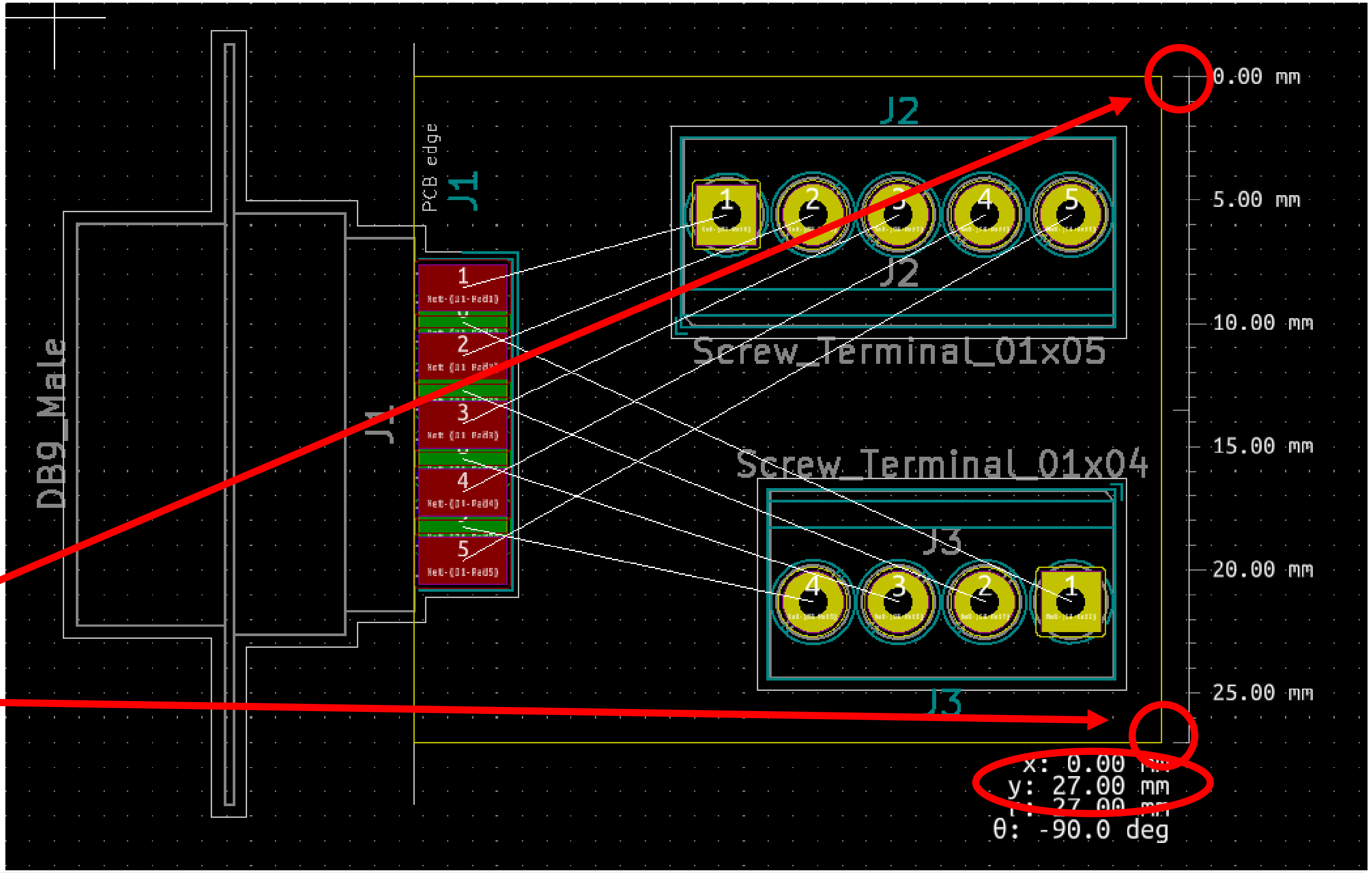
Measure X. Use the Caliper Tool.



Click the Start and End of the Measurement.

PCB Layout Editor

Measure Y. Use the Caliper Tool.



Click the Start and End of the Measurement.

KiCad – Work Flow

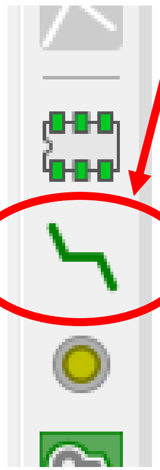
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate Footprint With Component
  - Generate Netlist
- PCB Layout Editor
  - Import Netlist
  - Mounting Holes
    - 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

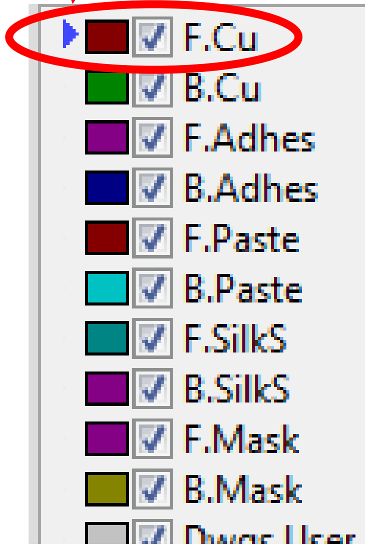
PCB Layout Editor

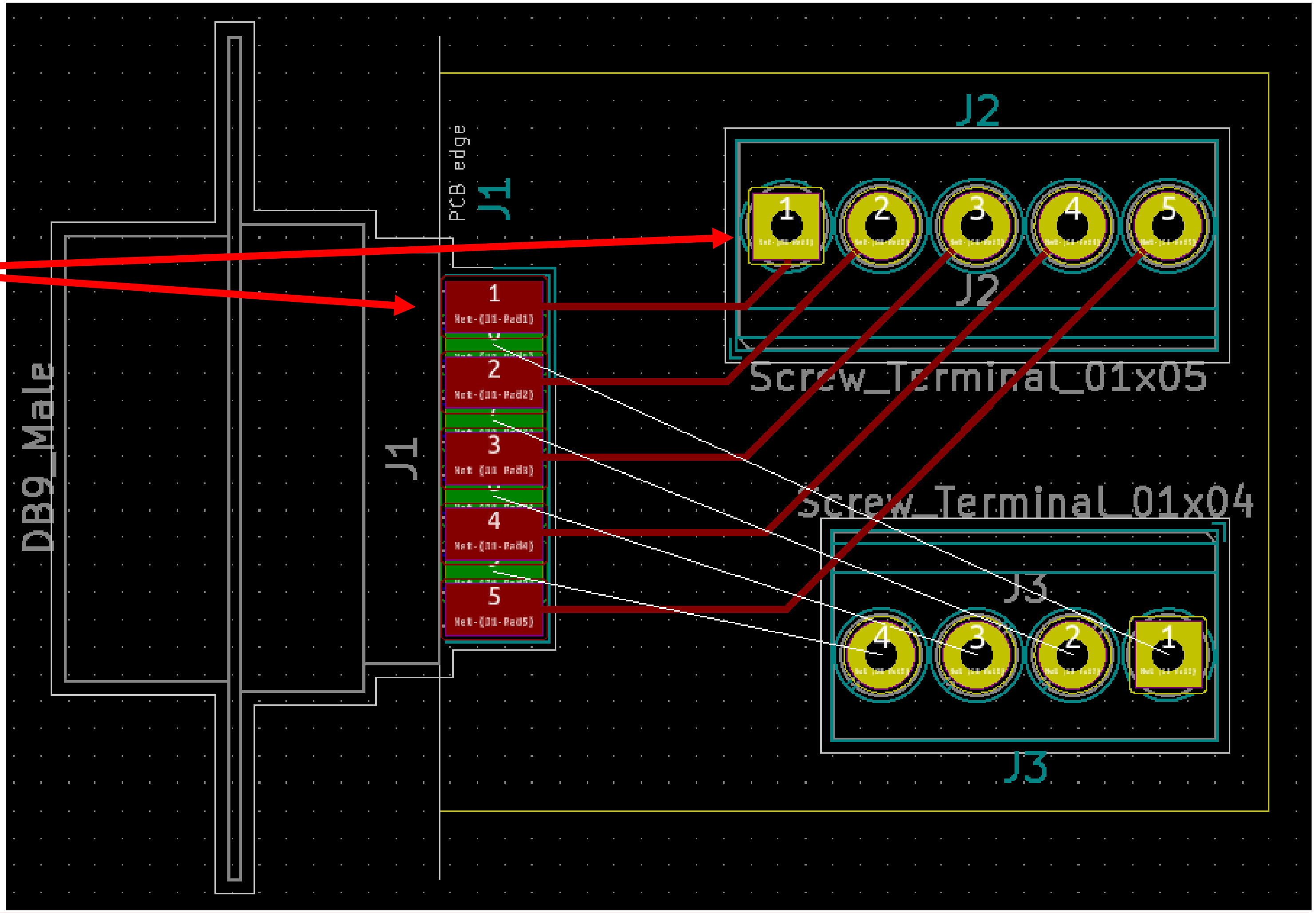
Click Route Track

Click Front Copper

Click Start and End of Trace.

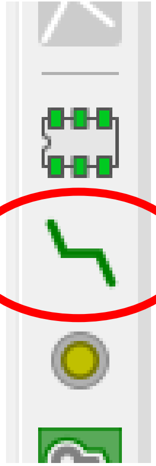


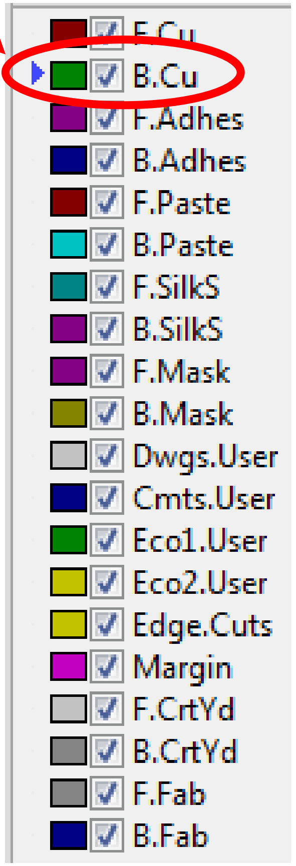


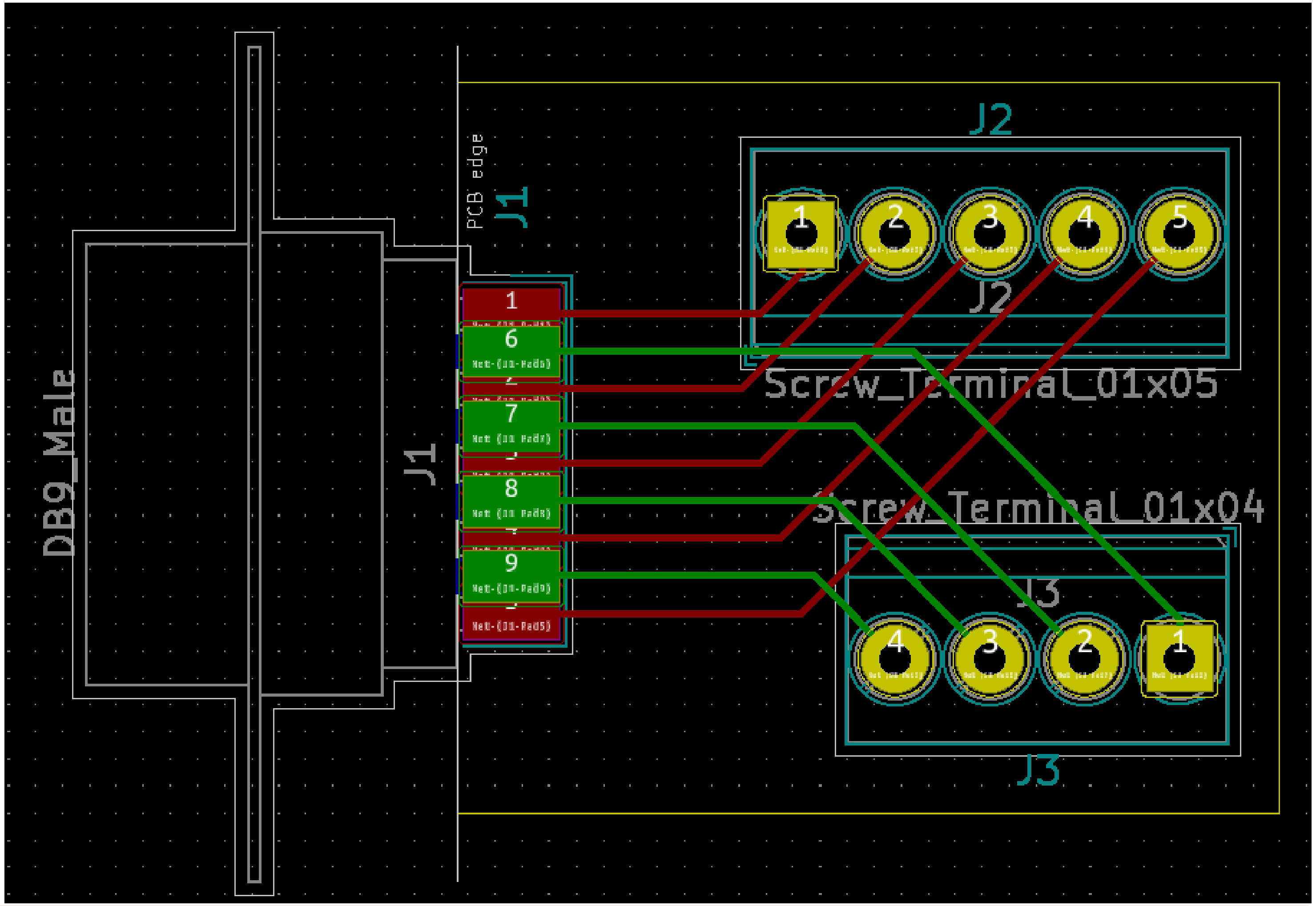


PCB Layout Editor

Now the Back







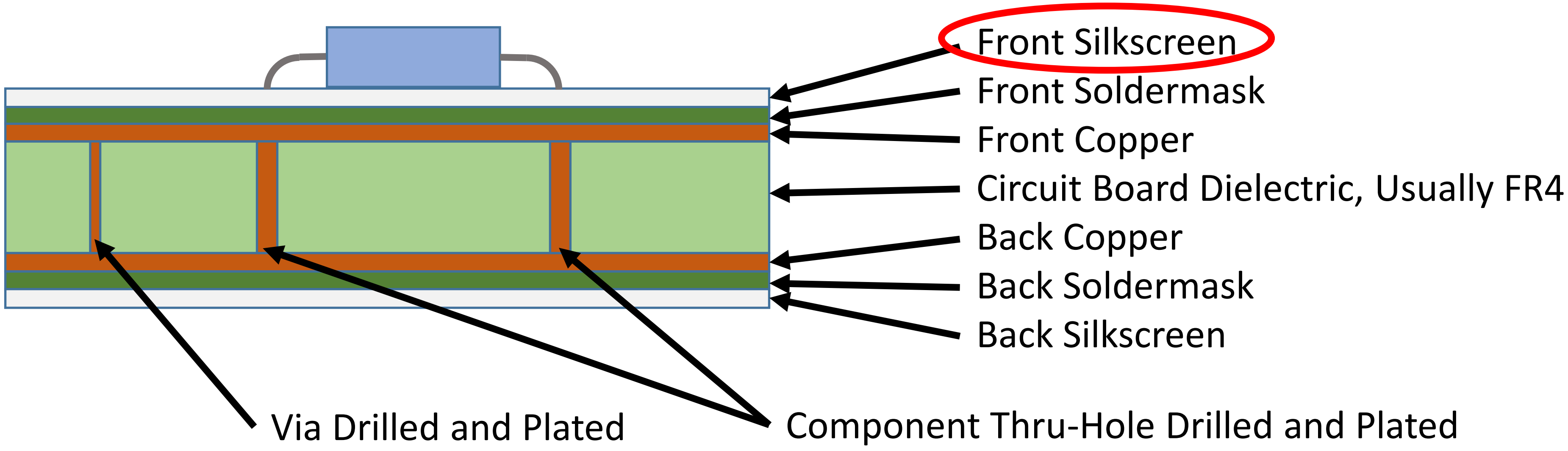
KiCad – Work Flow

- 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
  - Import Netlist
  - Mounting Holes
    - 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

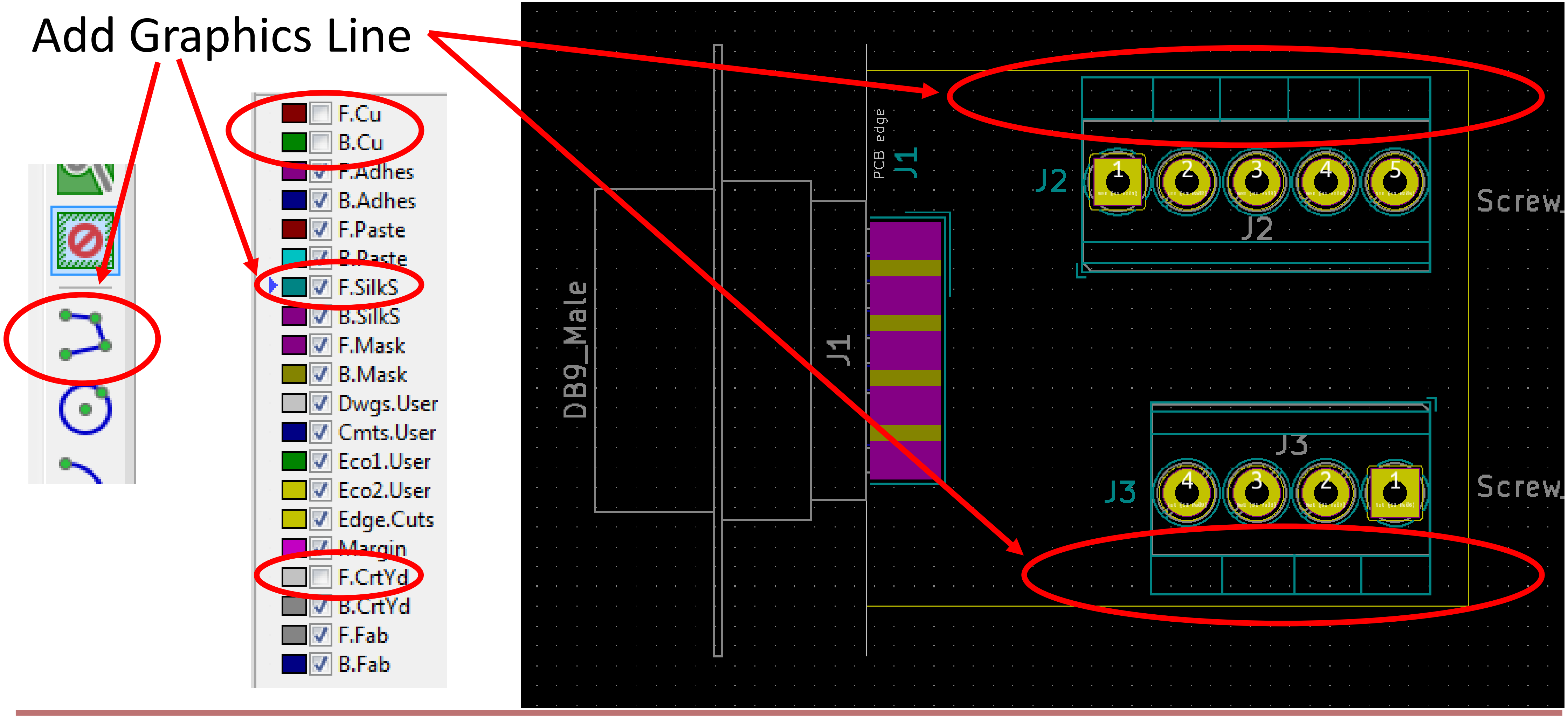
Anatomy of a PCB

- 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



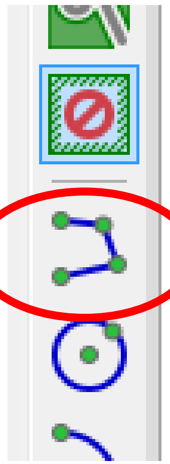
PCB Layout Editor

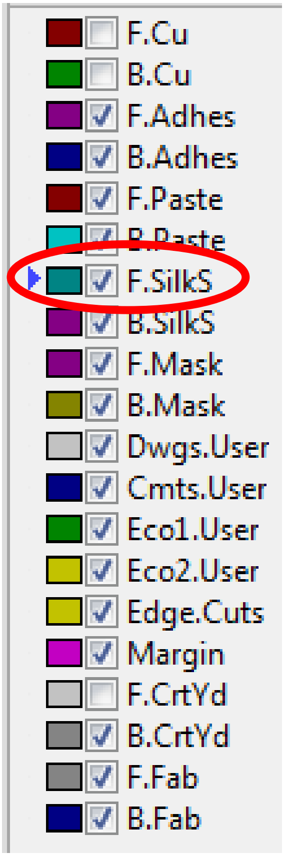
Add Graphics Line

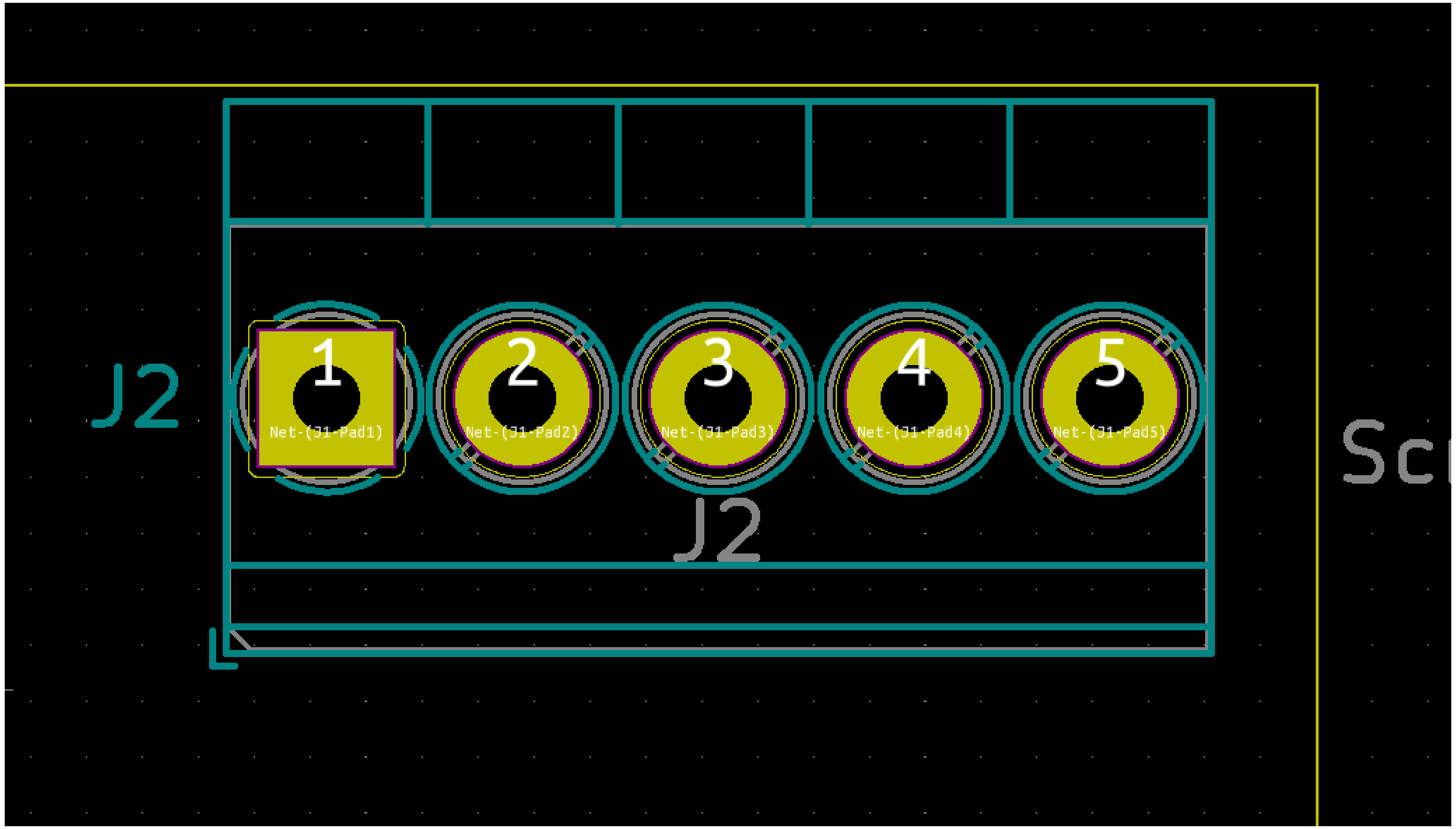


PCB Layout Editor

Add Graphics Line



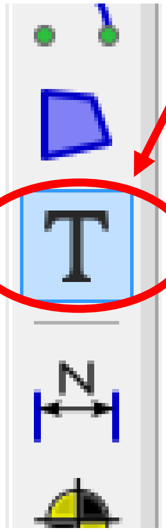


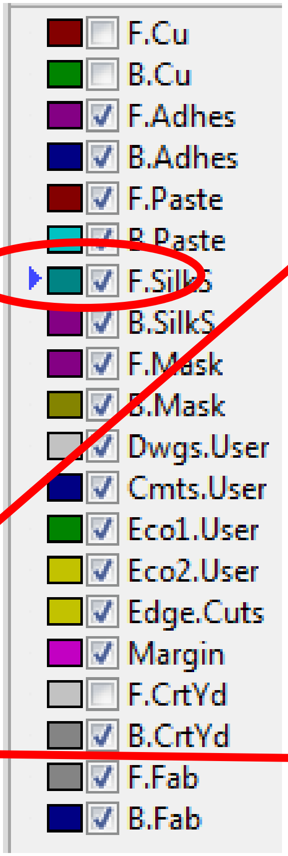


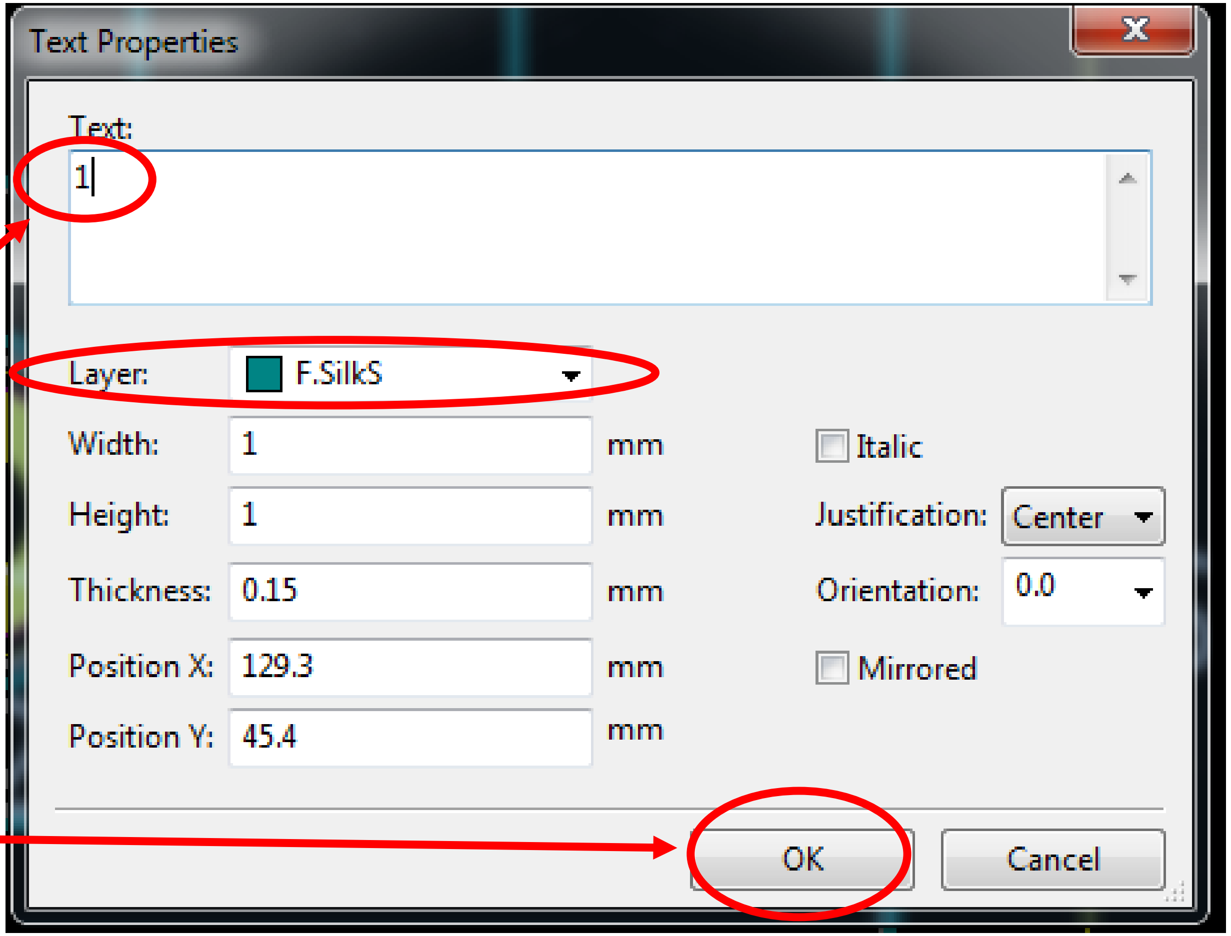
PCB Layout Editor

Click Add Text Icon

Click Sheet







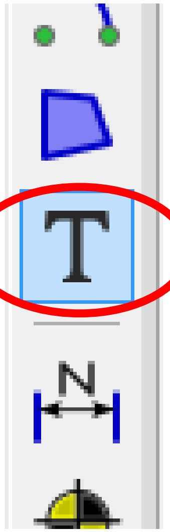
Add Text

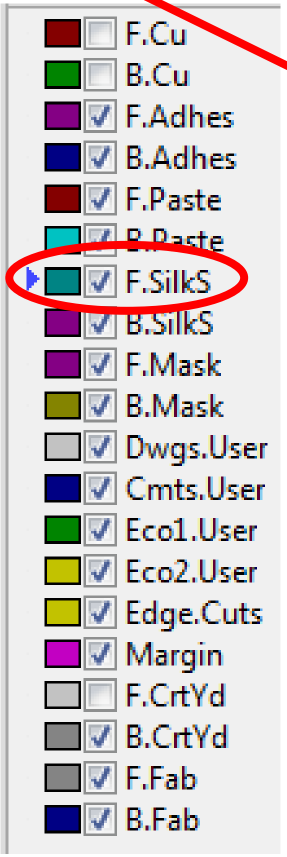
Click OK

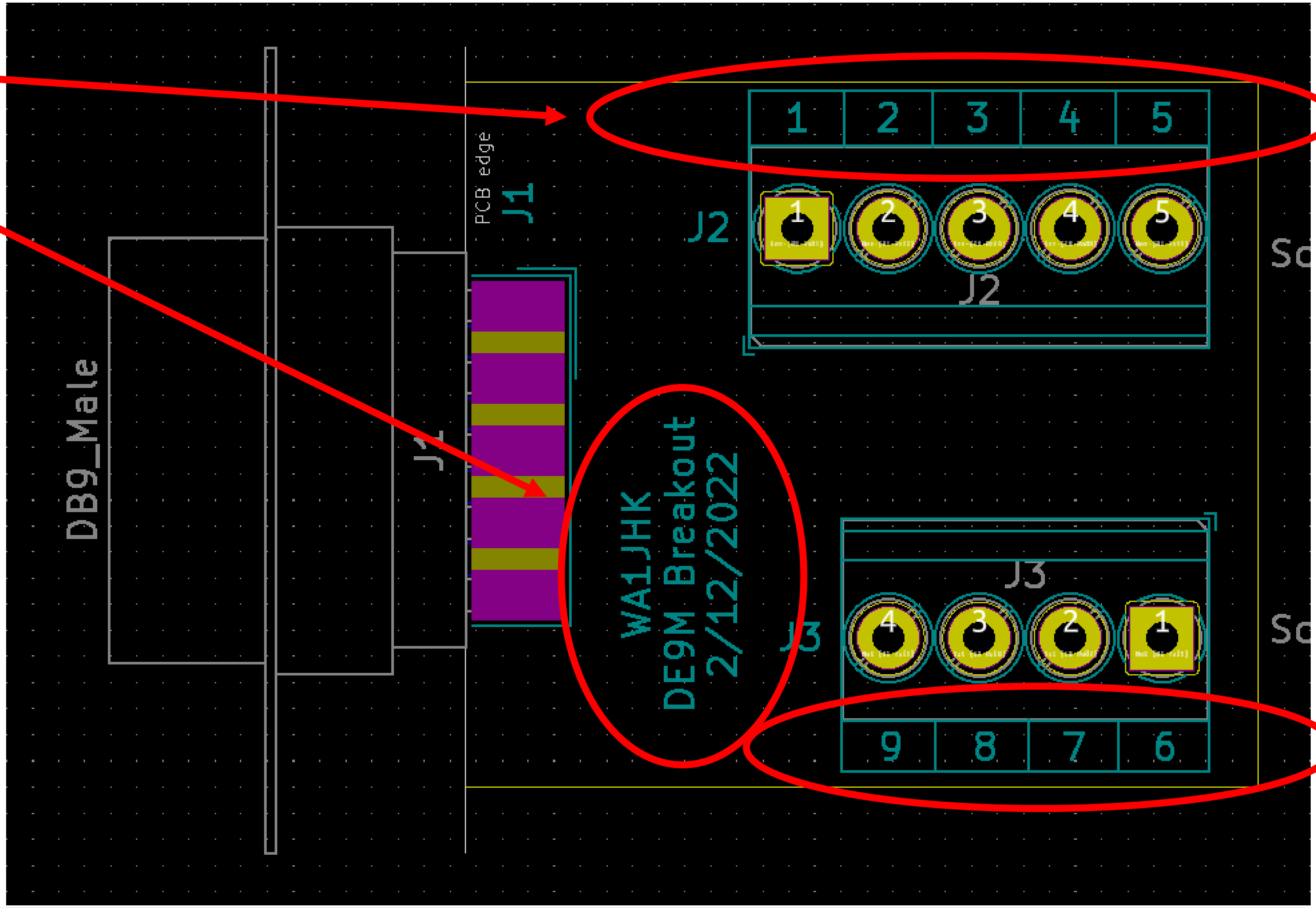
Click Sheet

PCB Layout Editor

Add Text

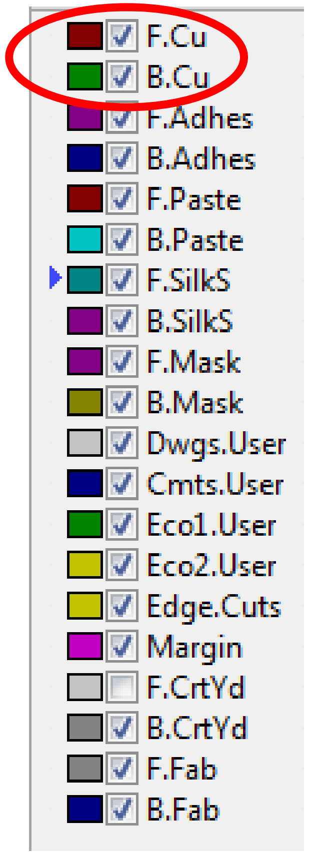


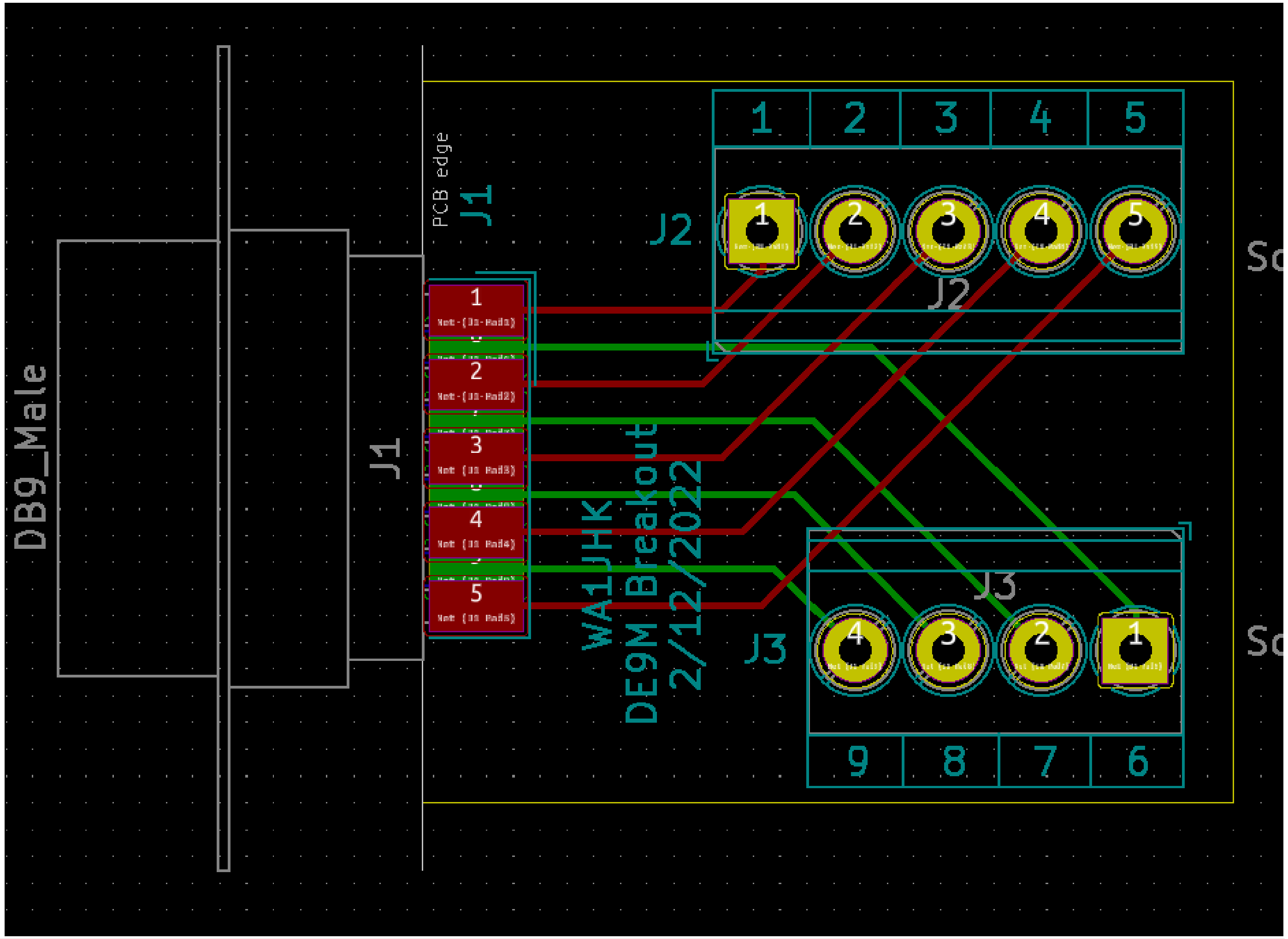




PCB Layout Editor

Copper Layers Back On



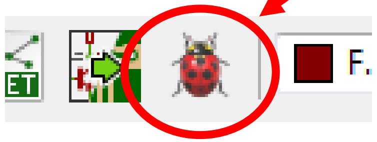


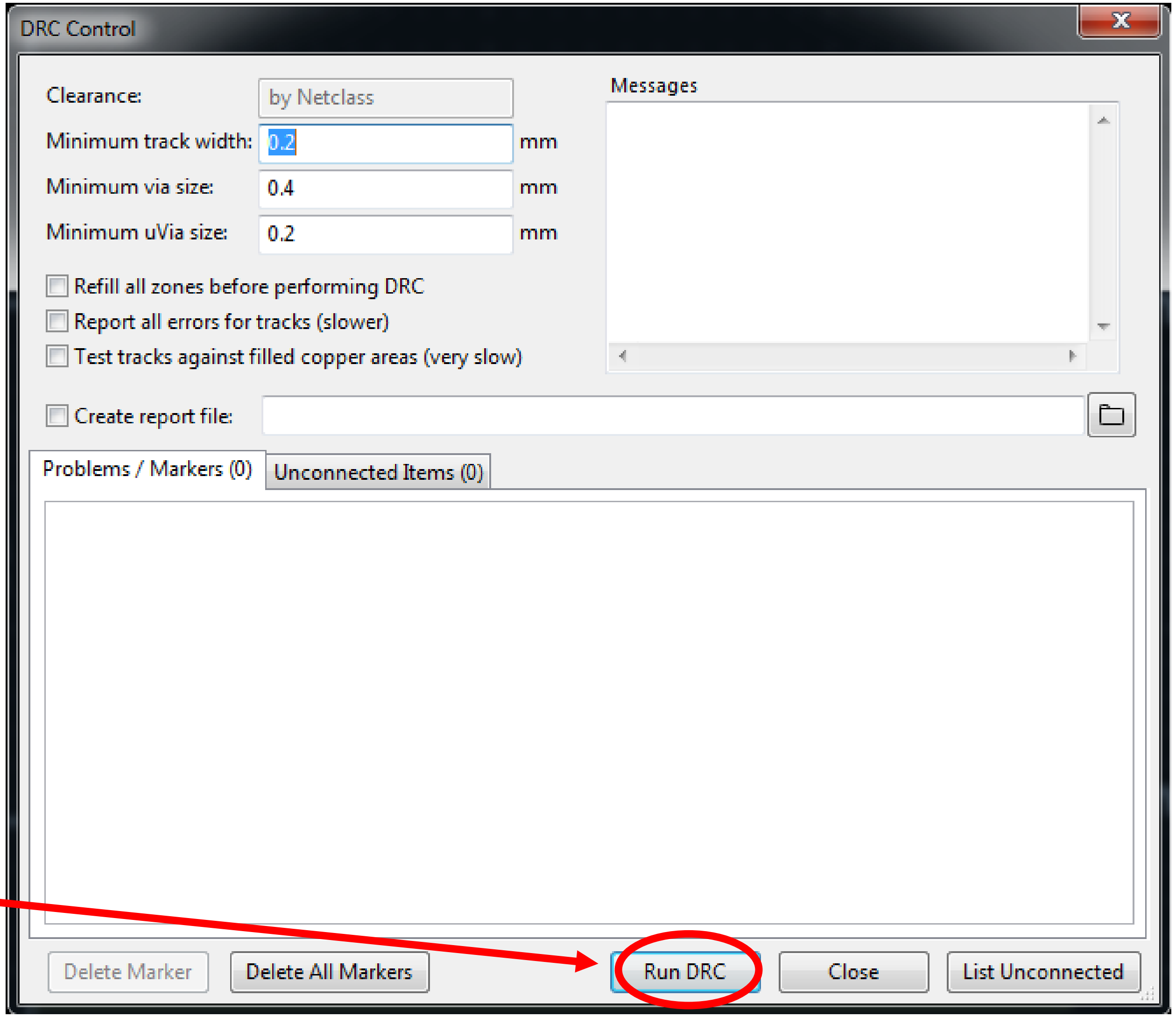
KiCad – Work Flow

- 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
  - Import Netlist
  - Mounting Holes
    - 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

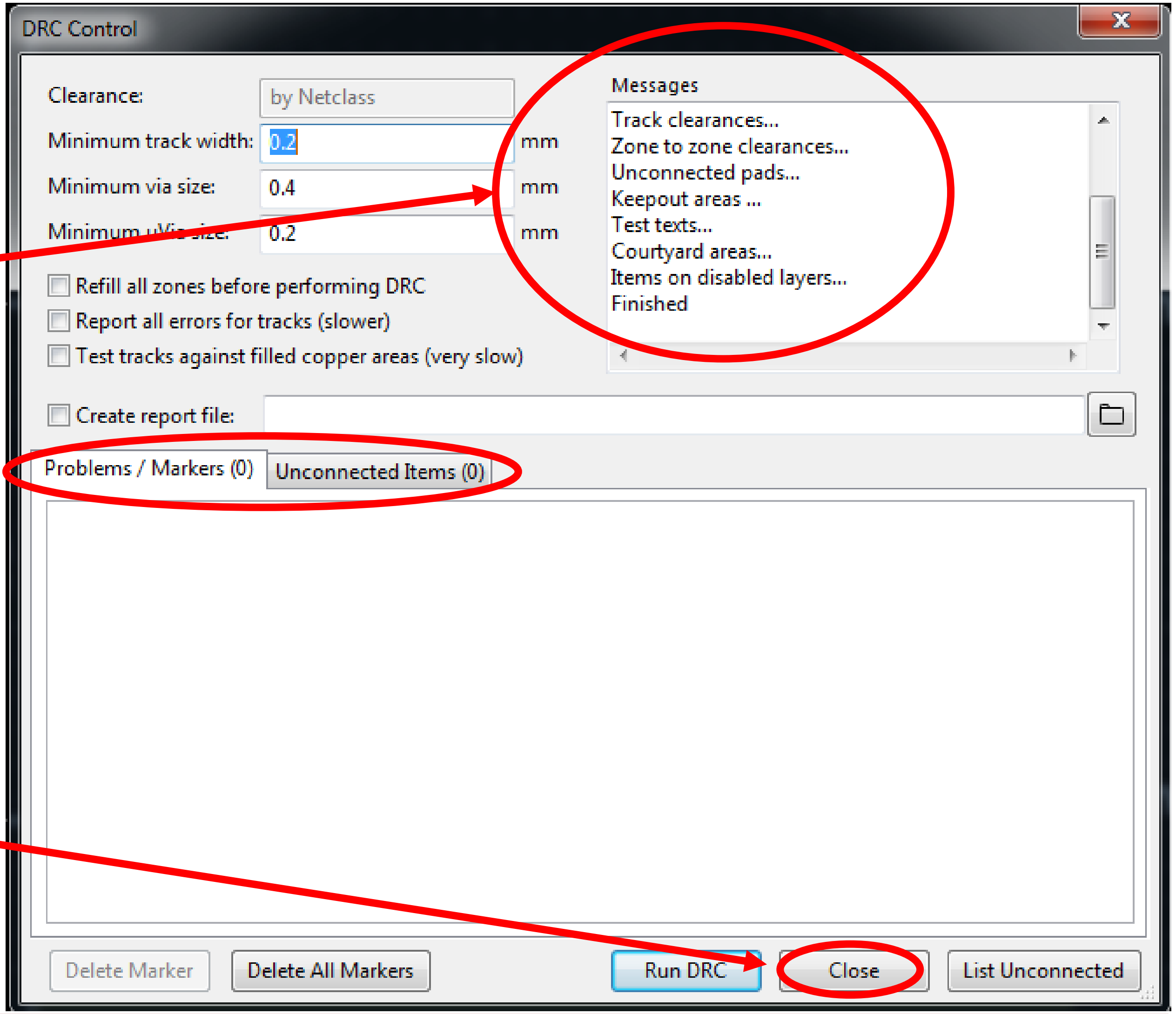
PCB Layout Editor

Review Messages

Look for “Finished”

Check Errors

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
  - Import Netlist
  - Mounting Holes
    - 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

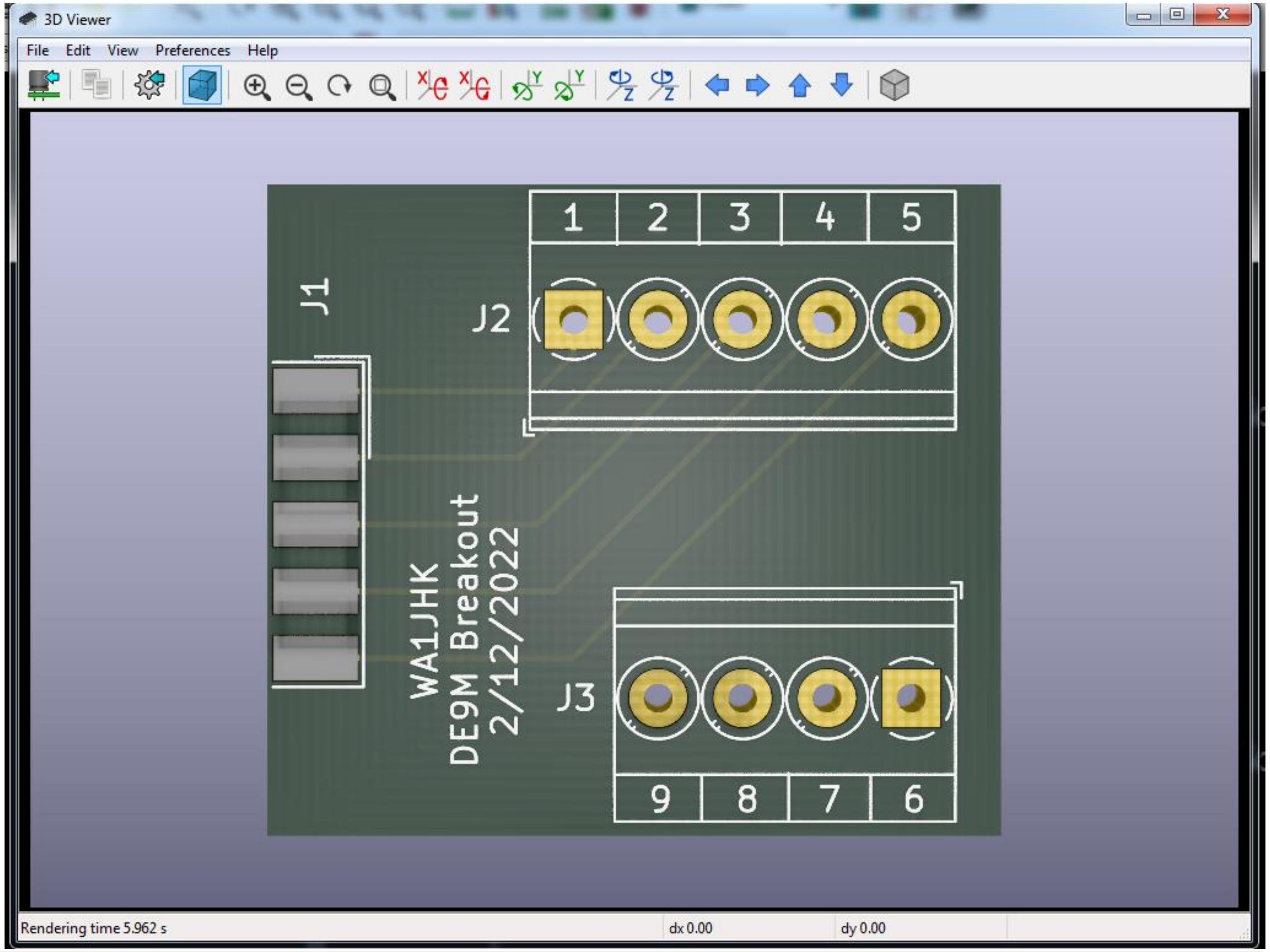
PCB Layout Editor

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.



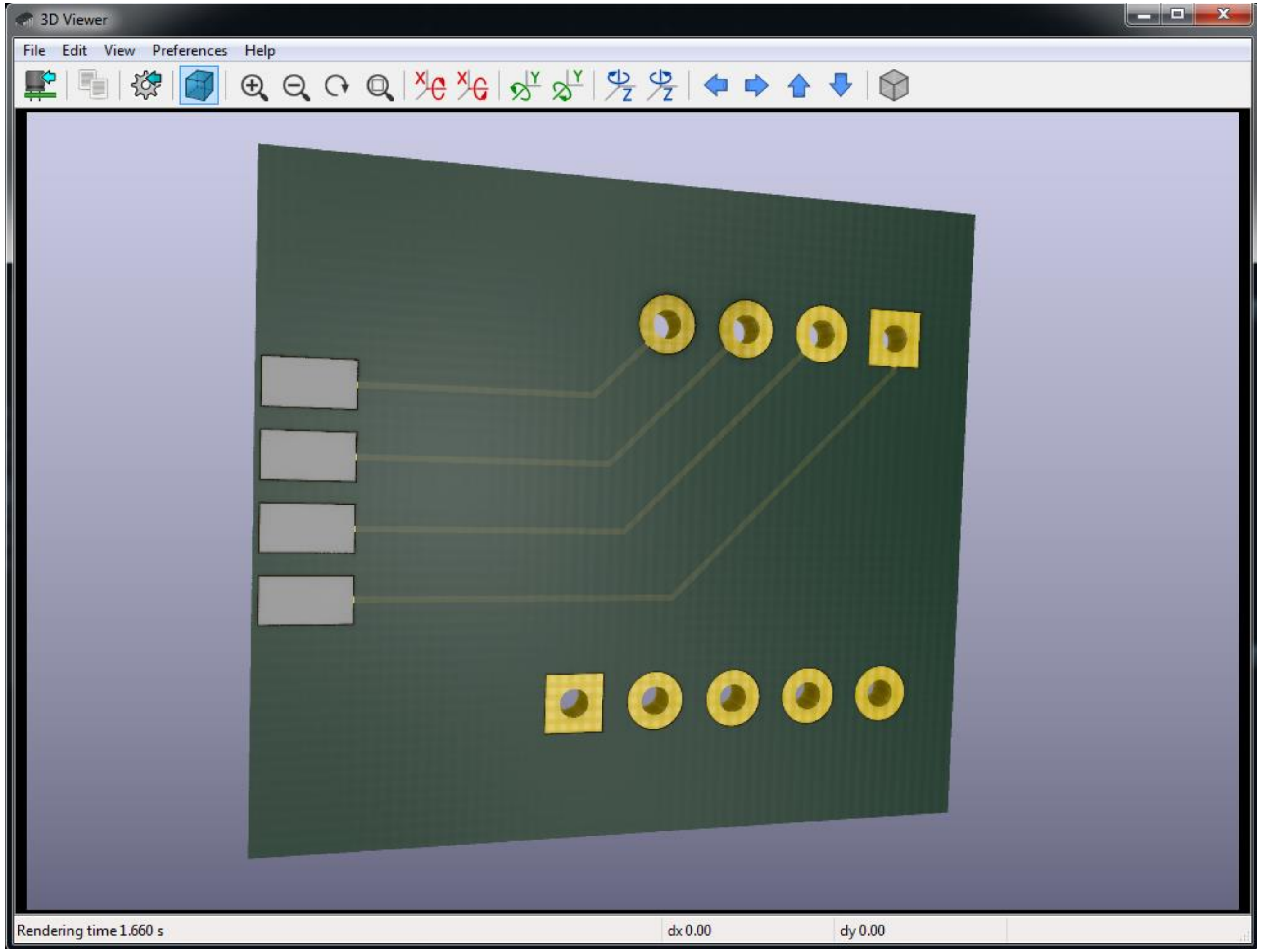
PCB Layout Editor

Press ALT-3

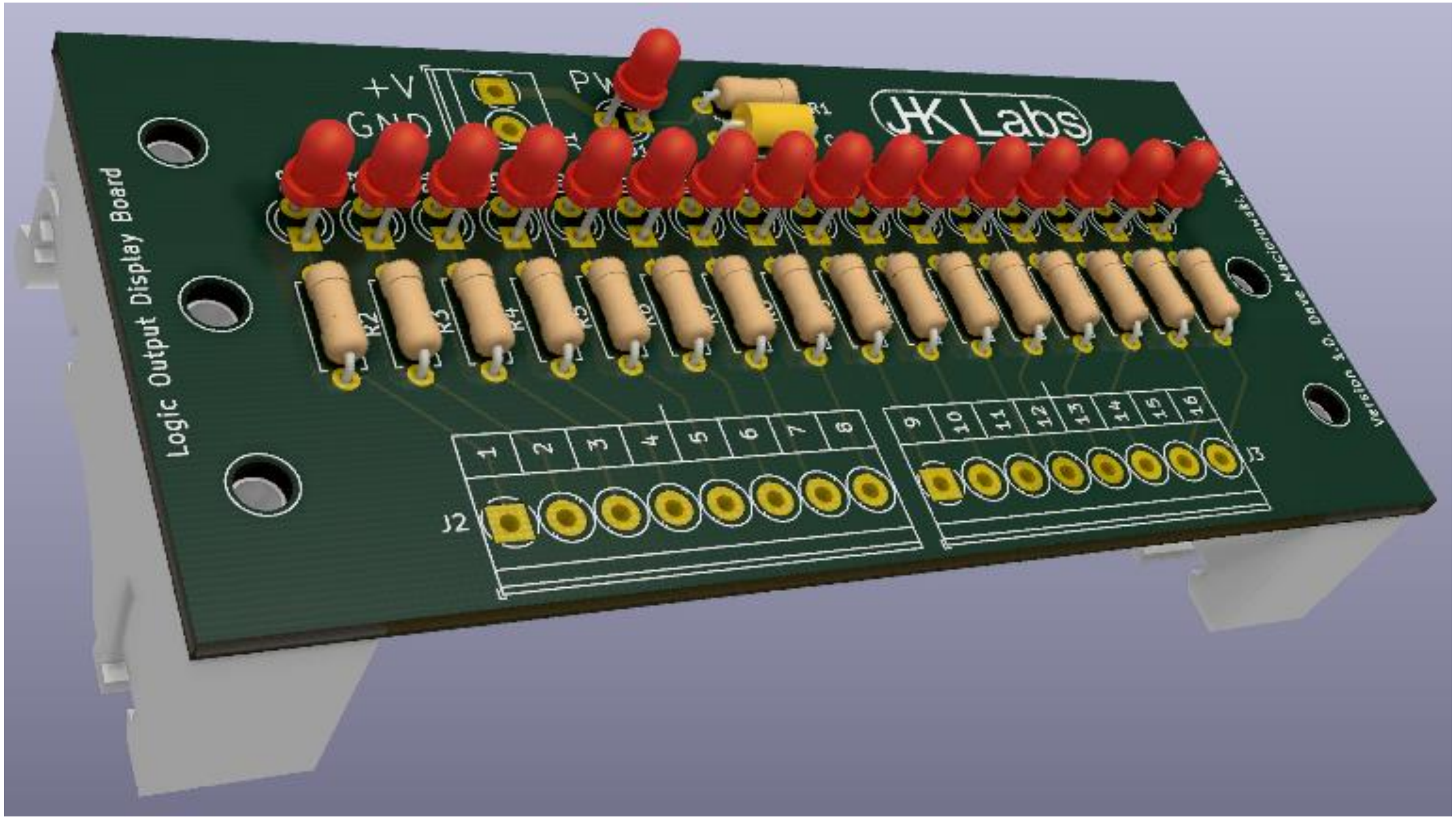
3D Viewer Displays in a new window.

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
  - Import Netlist
  - Mounting Holes
    - 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

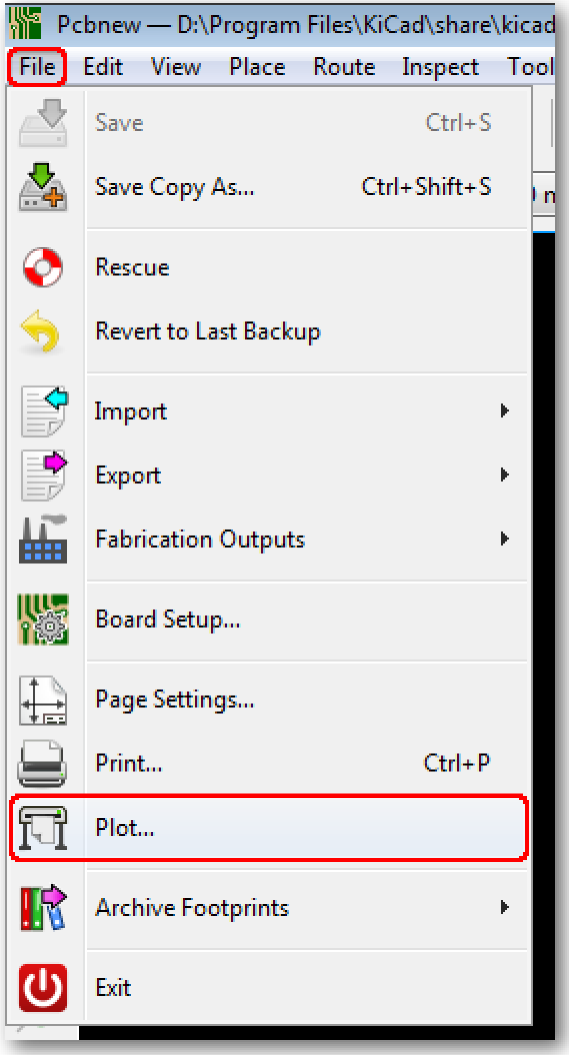
PCB Layout Editor

- Generate Gerbers and Drill Files
- Manufacturers Each Have Their Own Requirements
  - JLCPCB -- https://support.jlcpcb.com/article/149-how-to-generate-gerber-and-drill-files-in-kicad
  - OSHPARK -- https://docs.oshpark.com/submitting-orders/preorder-checklist/

PCB Layout Editor

Click File

then Plot...



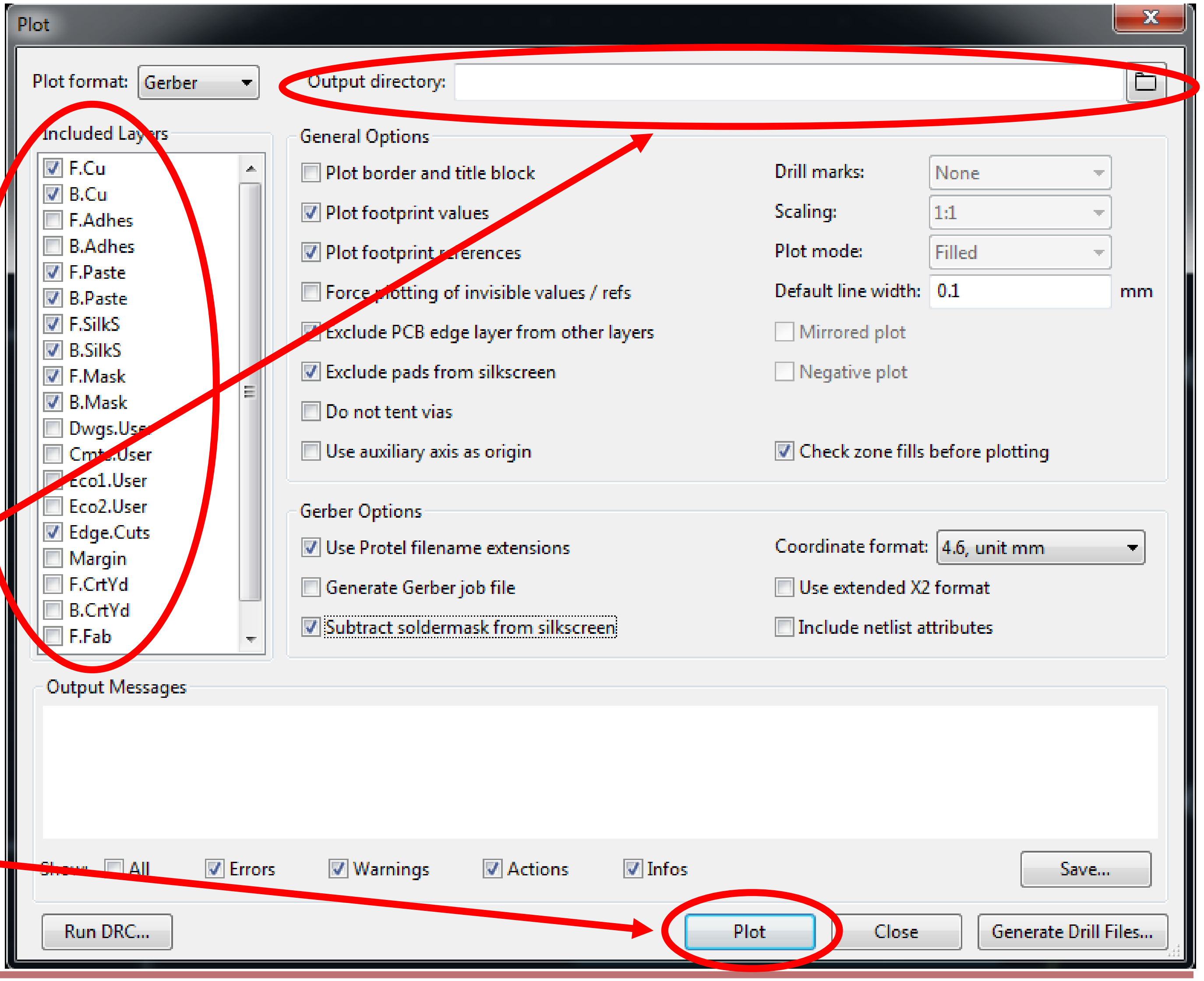
PCB Layout Editor

Included Layers Are Checked

Verify Checkboxes

Default Directory OK

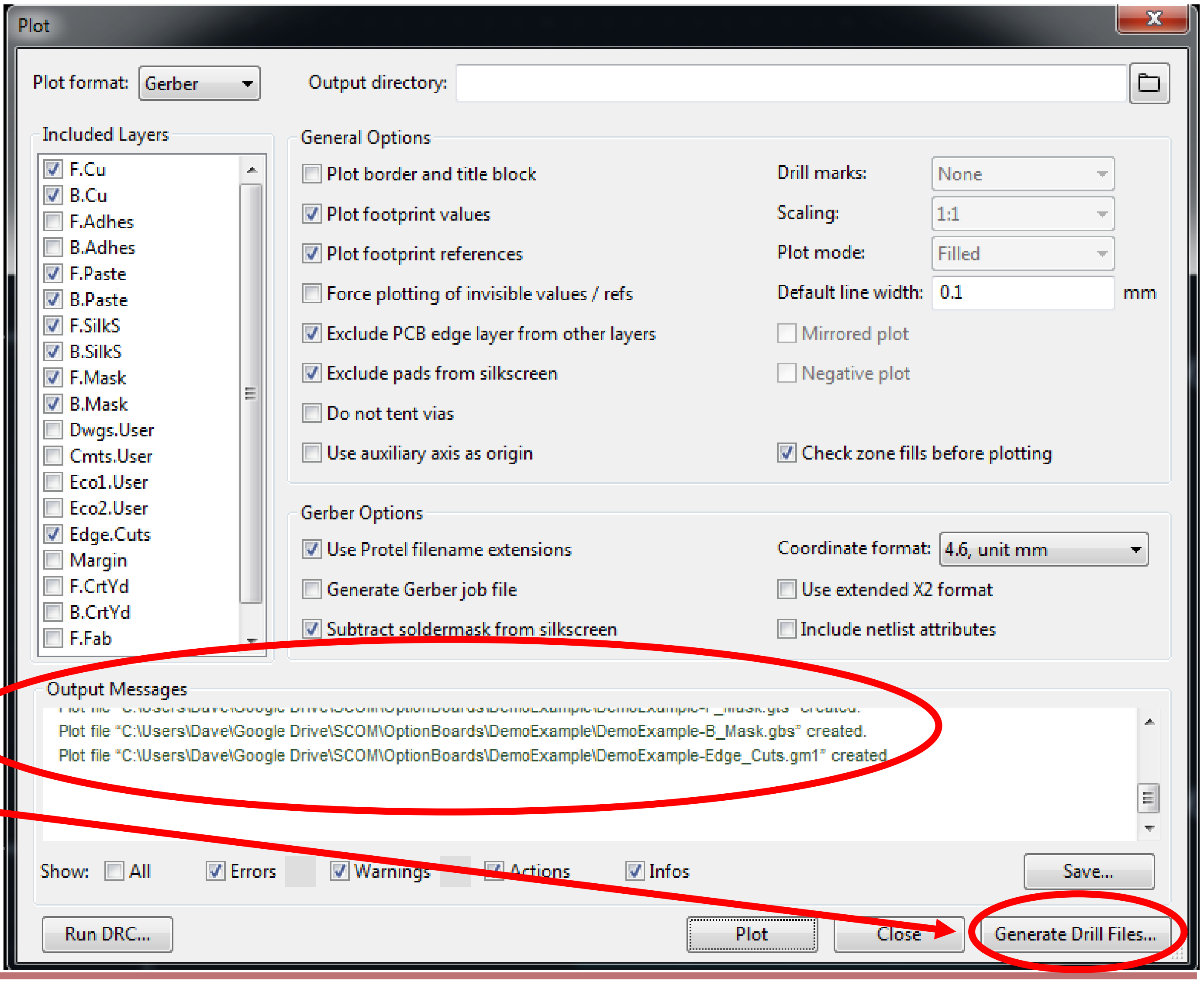
Click Plot



PCB Layout Editor

Output Messages Verify Files Generated

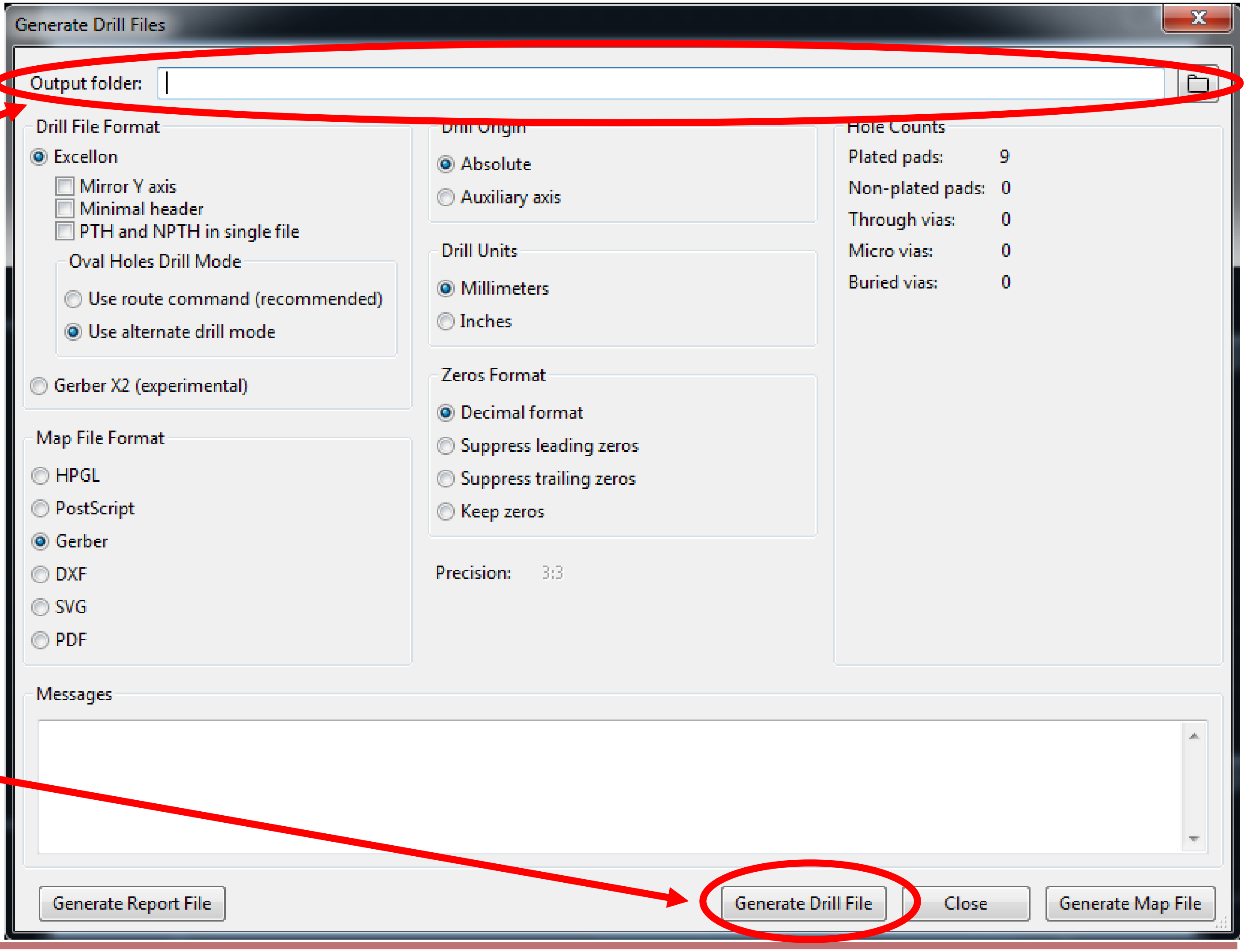
Click Generate Drill Files



PCB Layout Editor

Default Same Directory as Gerbers

Click Generate Drill File

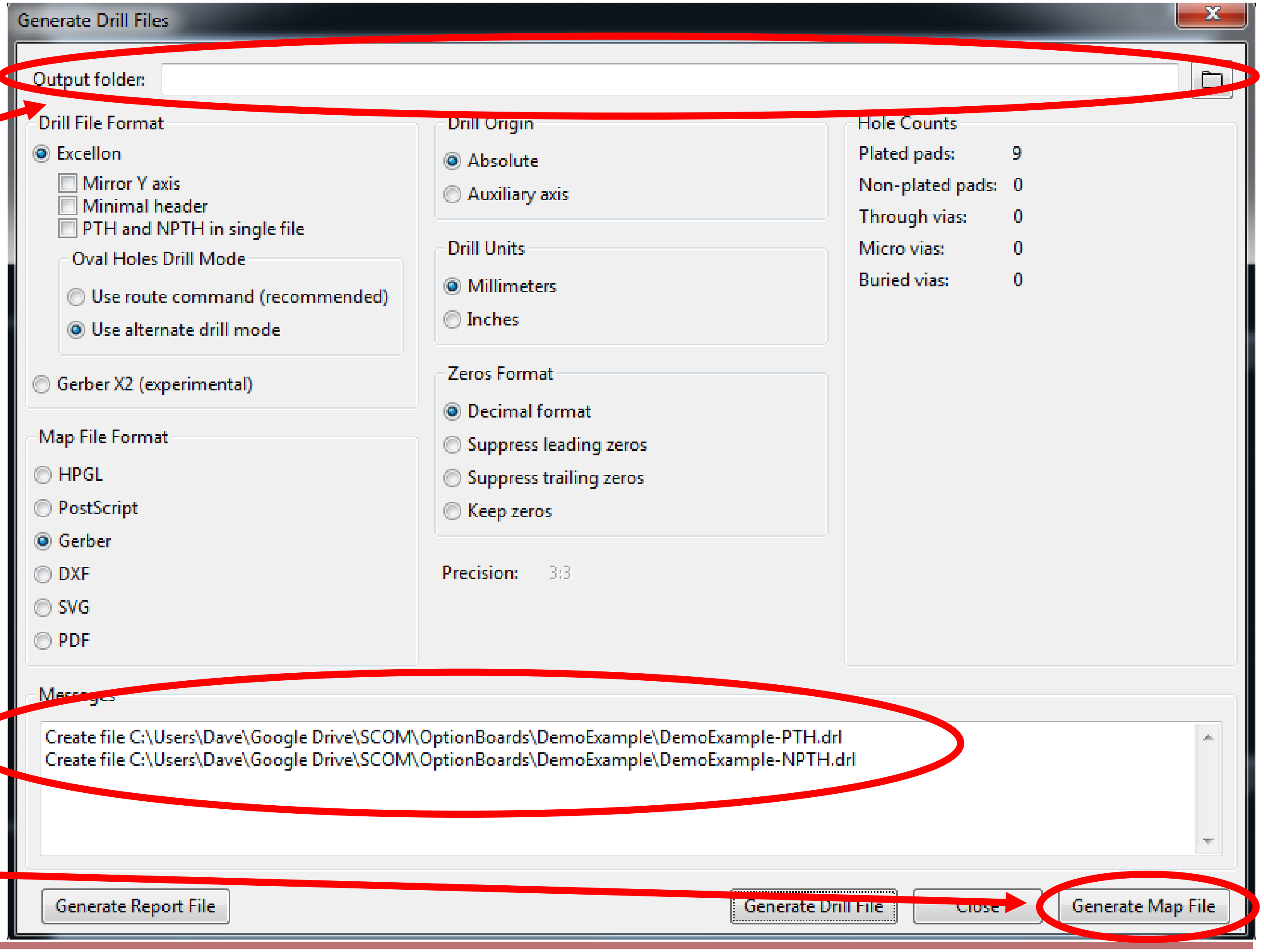


PCB Layout Editor

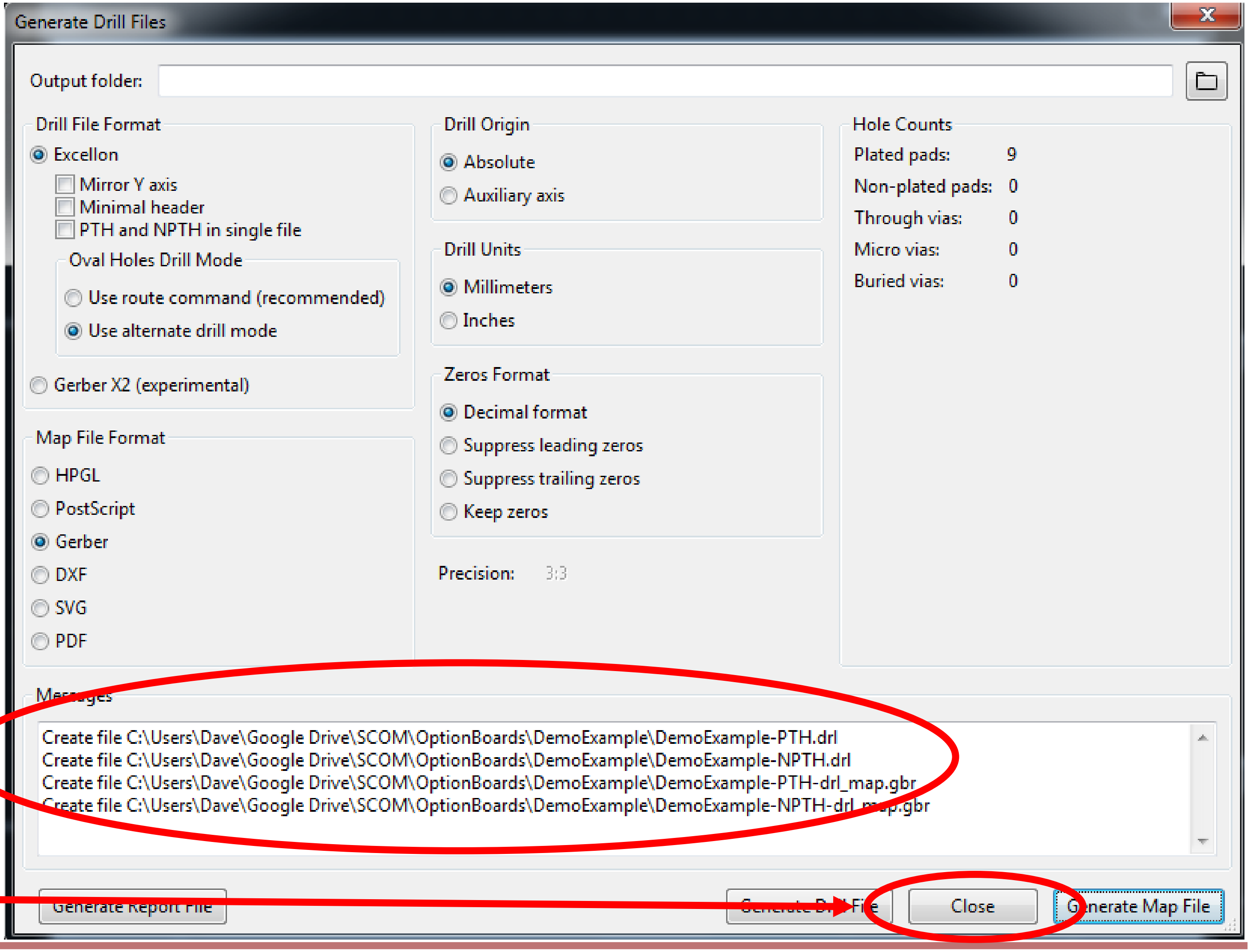
Default Same Directory as Gerbers

See Messages

Click Generate Map File



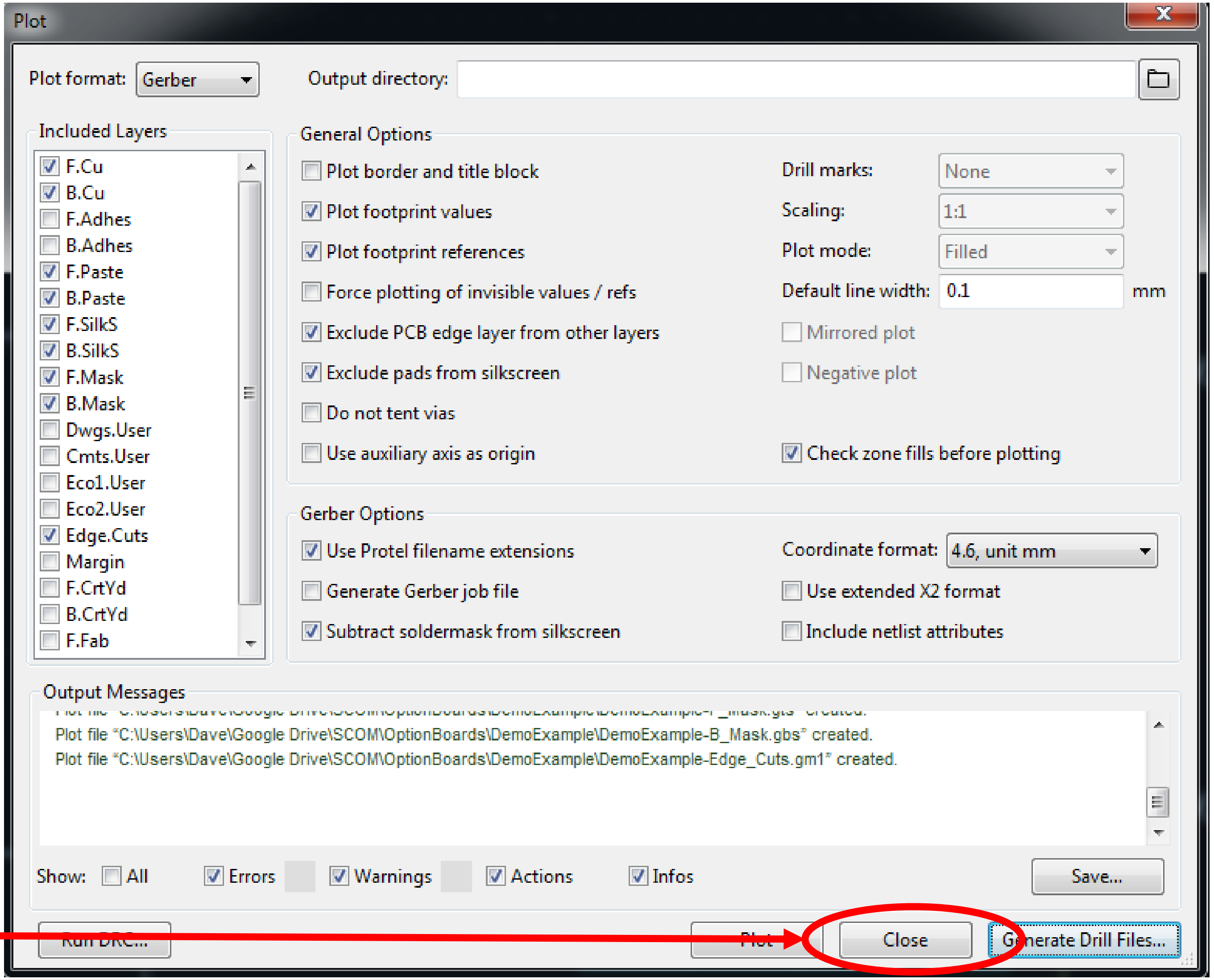
PCB Layout Editor



See Messages

Click Close

PCB Layout Editor

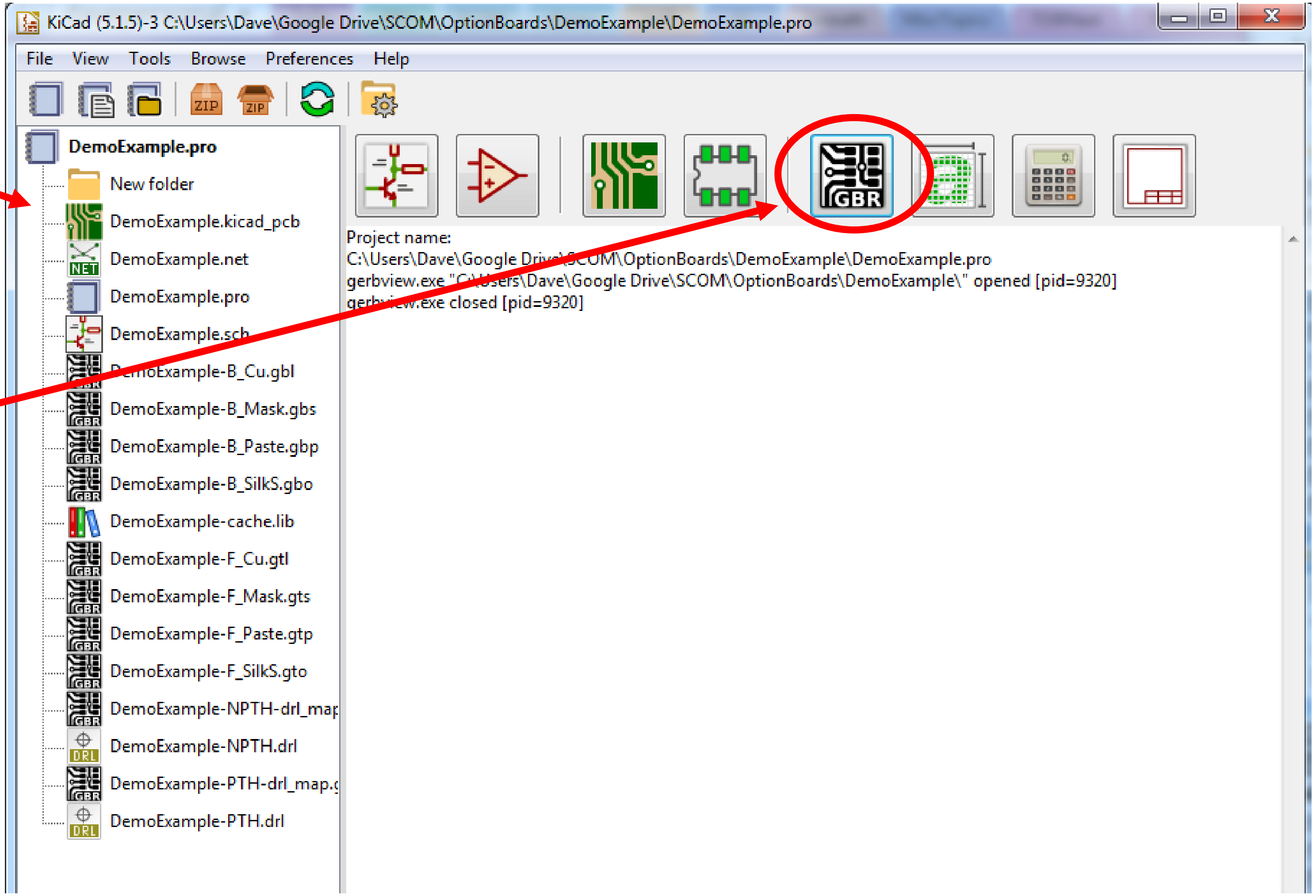


Click Close

KiCad – Main

Output Files

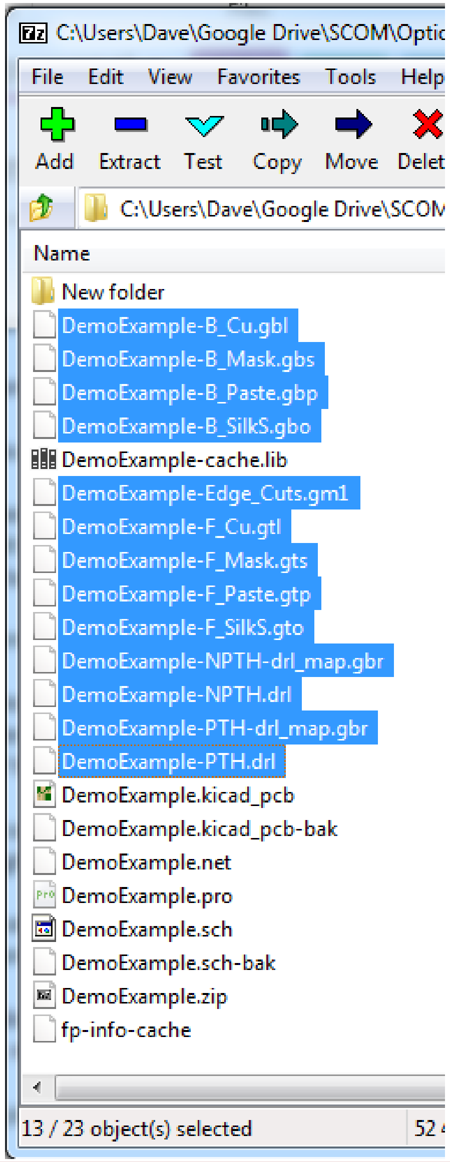
Gerber Viewer Looks at Output Files



Zip Gerbers and Drill

Select the Files and Zip them Up

DemoExample_Gerbers_220212.zip



Order Your Boards!!!

Browse to Jlcpcb.com

Sign In/Create Account

Upload your .zip file

Review Quote.

Order!

References

- KiCad Website
  - https://www.kicad.org/
- KiCad Tutorials
  - https://www.youtube.com/c/JohnsBasement
  - https://www.youtube.com/c/contextualelectronics
  - https://learn.sparkfun.com/tutorials/pcb-basics/all
- JLCPCB Reference
  - https://support.jlcpcb.com/article/149-how-to-generate-gerber-and-drill-files-in-kicad
- Misc
  - https://en.wikipedia.org/wiki/Gerber_format

Questions?

THANK YOU!!!