KiCad QuickStart

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

JHK Labs

References

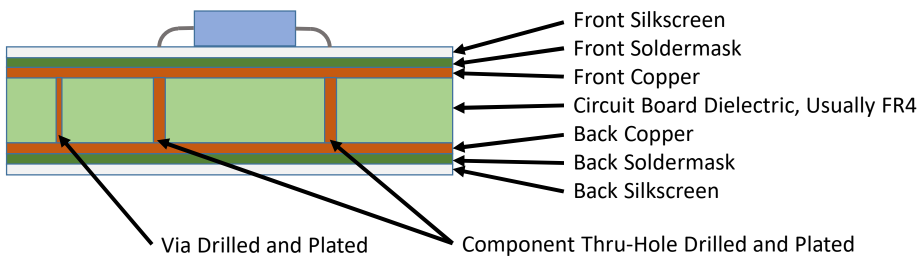
- KiCad Website
  - https://www.kicad.org/
- KiCad Tutorials
  - https://www.youtube.com/c/JohnsBasement
  - https://www.youtube.com/c/contextualelectronics
  - https://learn.sparkfun.com/tutorials/pcb-basics/all
- JLCPCB Reference
  - https://support.jlcpcb.com/article/149-how-to-generate-gerber-and-drill-files-in-kicad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



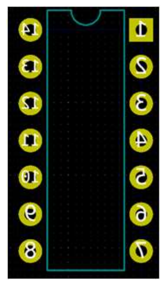
This is what a circuit board looks like from the side. KiCad needs to generate the Gerber files used in the manufacturing process. In addition to what the copper, silkscreen, and soldermask look like, there’s a drill map of all the holes that need to be drilled and what size they are. There are also instructions about whether the holes need to be plated between the top and bottom or left disconnected.

This diagram shows two layers of copper, a front and back layer. But circuit boards can be built of thinner layers alternating copper and the board core material, up to 30 layers.

Anatomy of a PCB

- Schematic Symbols

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



- Resistors
  - .3 Inch Axial

  - 0603 == .06 x .03 Inch

We are going to create a Schematic that shows the wiring of our board. We need Symbols. KiCad has a very large library of symbols that you can choose from. And, you can create your own. Some common ones are shown here.
Each Symbol on the Schematic has an Associated Footprint. There are Footprint Libraries as well. For example, the Dual Inline Package was the standard for ICs for many years, then Surface Mount Packages came along. The same for resistors.

KiCad – Work Flow

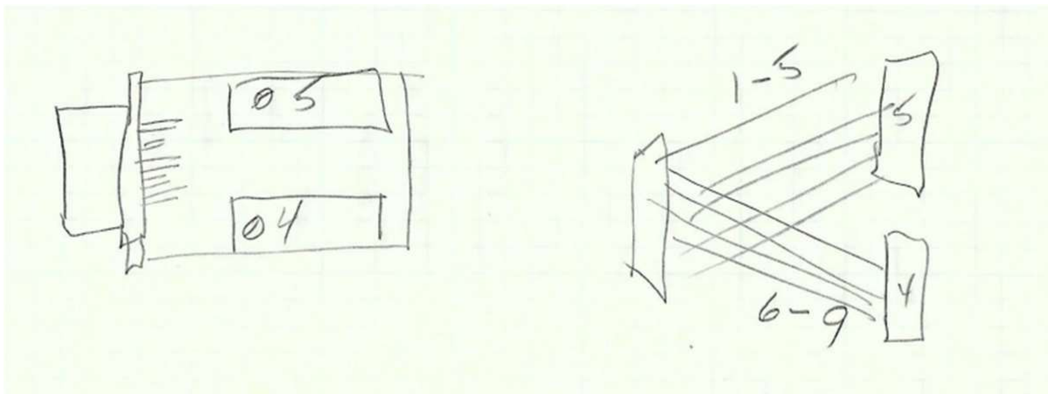
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate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

This is the Work Flow that I generally use to produce a design. Think of it as a checklist. If you skip a step, the tools will make you go 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

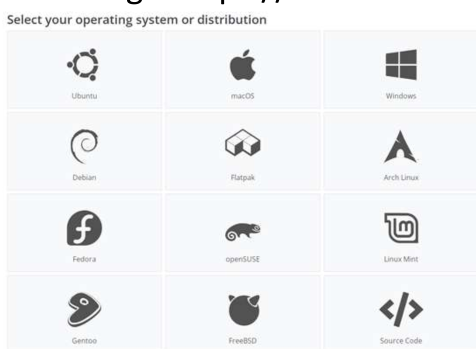
Example Project – Hand Sketch



The example project is a DE9 breakout board. The left sketch is a physical layout. The right sketch is a rough schematic. We'll come back to this later.

KiCad

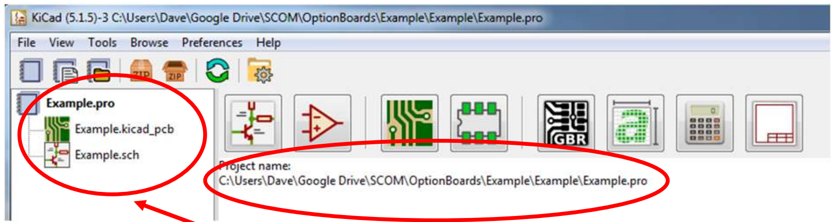
- Pronunciation – KEY-CAD
- Official Site -- https://www.kicad.org/
- Installing-- https://www.kicad.org/download/



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

The name of this tool suite is called KEY CAD.
You can load this tool suite on most platforms.
Today's examples are from Windows.

KiCad – Main Screen



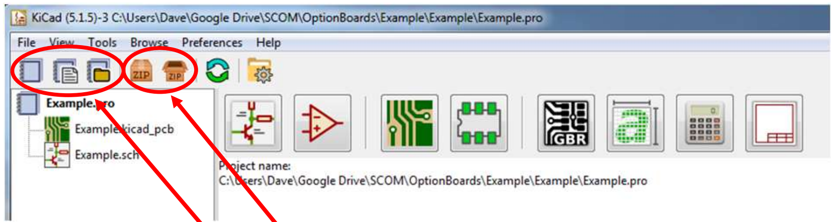
Project Folder Location

Project Files

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 10

On the left is a list of the project files. In the right is the project name with the full file path.

KiCad – Main Screen

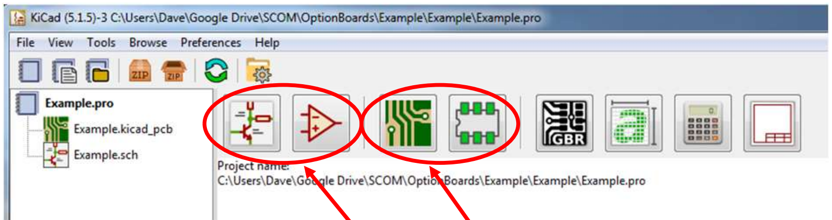


Archive and Unarchive Project

Create and Open Project Files

The left circle of tools are used to create and open project files. We're going to create a project file in a minute. The right circle of tools are used to archive and unarchive project files using Zip.

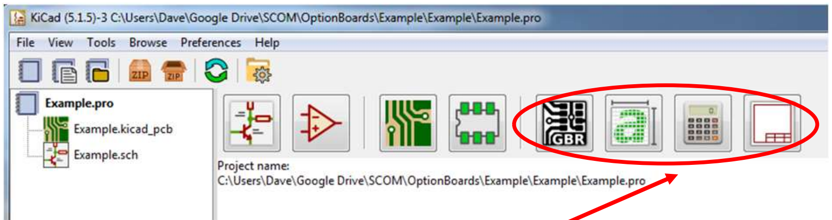
KiCad – Main Screen



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

We’ll use the Schematic Tools shortly to create the schematic for our project. Later we’ll use the PCB Tools to lay out a circuit board.

KiCad – Main Screen



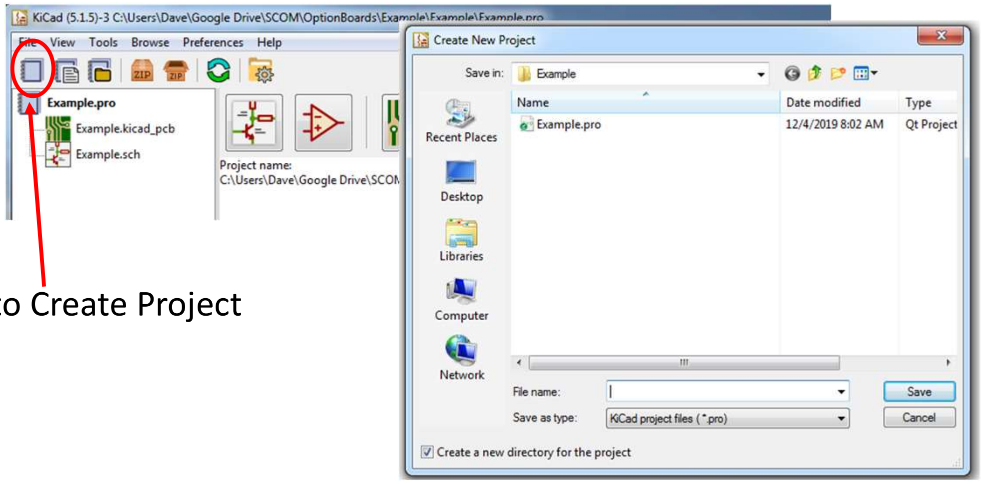
- Gerber Viewer
- Bitmap to Component Converter
- PCB Calculator
- Page Layout Editor

The Gerber Viewer looks at the manufacturing files that we'll send to the circuit board manufacturer. The other tools help with your design.

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



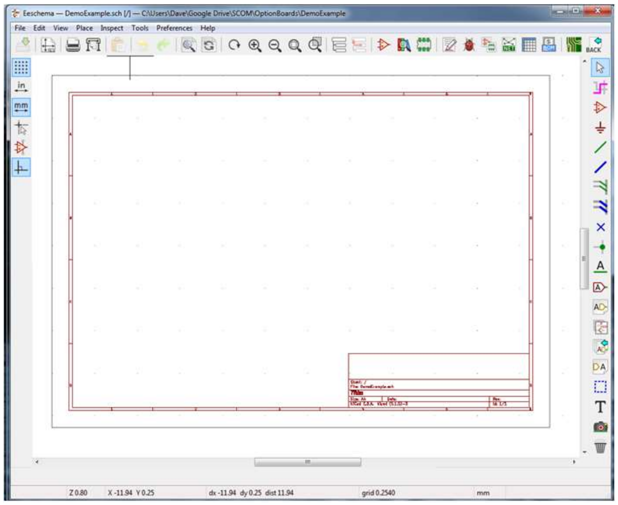
Click to Create Project

Click the Create Project Tool. In the dialog, provide a project name. Note the checkbox at the bottom. A new directory can be created to contains the project files.

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

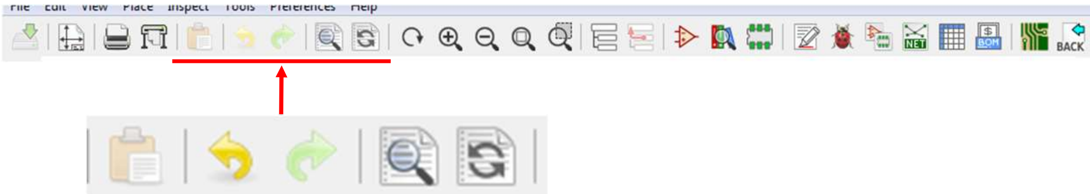
Left Toolbar is used for Mode Selection.
Top Toolbar contains Viewing and Navigation tools.
Right Toolbar contains Drawing Tools

Schematic Layout Editor

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

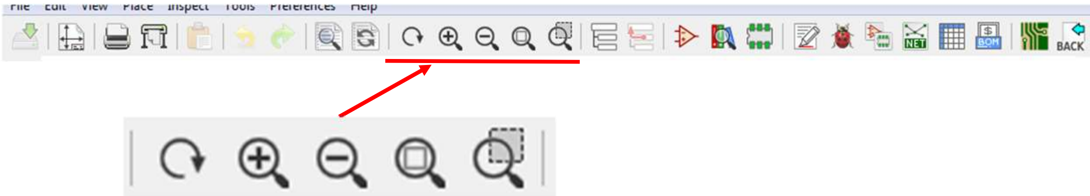
Looking at the Top Toolbar

Schematic Layout Editor



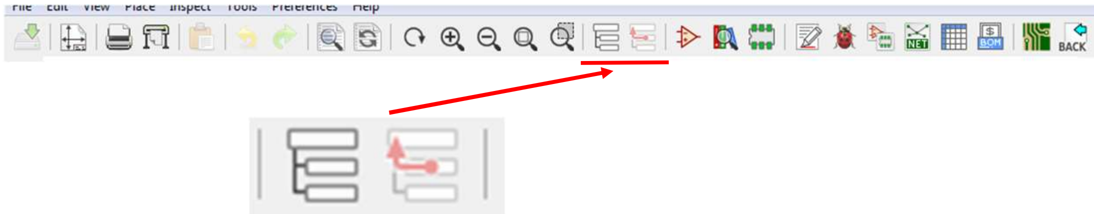
- Editing
  - Paste -- CTRL-V
  - Undo – CTRL-Z
  - Redo – CTRL-Y
  - Find Symbols and Text – CTRL-F
  - Find and Replace Symbols and Text

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

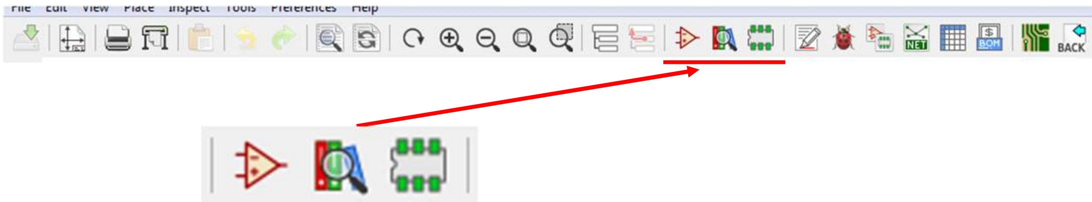
Schematic Layout Editor



- Multiple Schematic Sheets
  - Navigate Schematic Hierarchy
  - Leave Sheet

These are to move around in multiple schematic sheets. We aren't going to use these today.

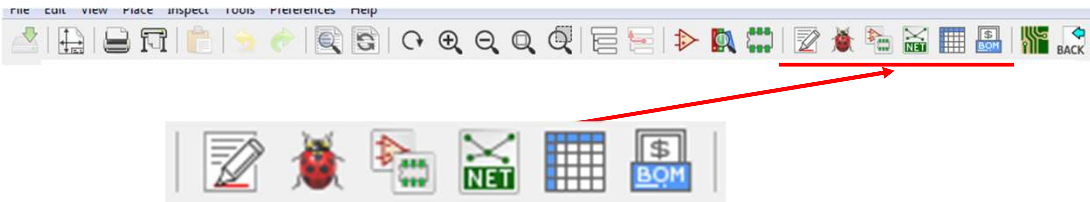
Schematic Layout Editor



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

These tools let you work with Symbols and Footprints.
I recommend that you take some time to browse the Symbol Libraries. The libraries are very extensive.

Schematic Layout Editor

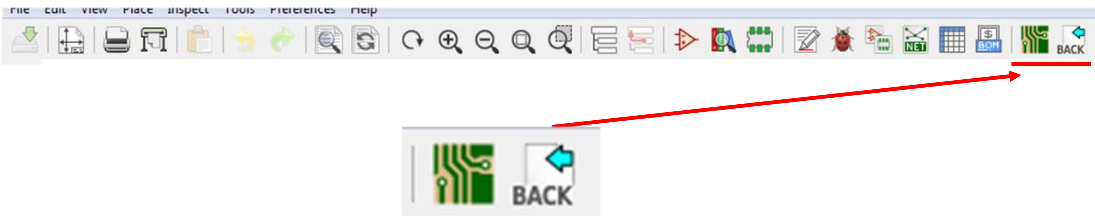


- Process Schematic
  - Annotate Schematic Symbols – Adds Reference Designators
  - Perform Electrical Rules Check – Find Errors that Would Prevent Netlist
  - Assign PCB Footprints to Schematic Symbols – Resolve Errors from Check
  - Generate Netlist – Will be used by PCB Layout Editor
  - Edit Symbol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

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 23

There are tools to process the schematic that you enter. We'll be using several of these today.

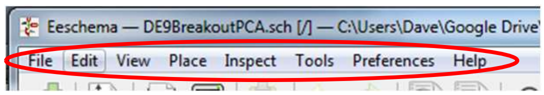
Schematic Layout Editor



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

Once we have the schematic entered and processed, we'll be ready to start the PCB Layout Editor using this icon.

Schematic Layout Editor



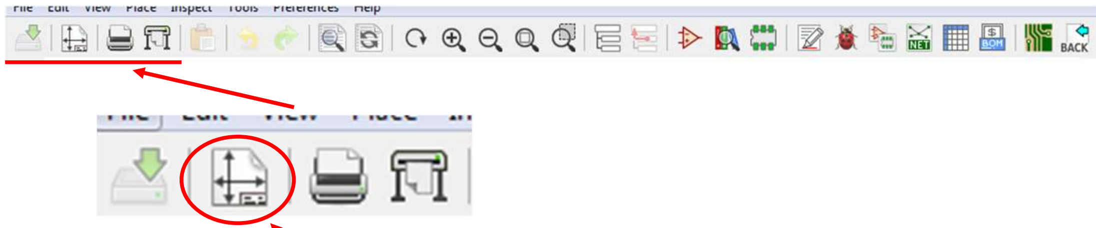
- Menu
  - File Management
  - Editing
  - Alternate Way to Access Tools
  - Preferences
  - Help – List Hotkeys

There is the usual menu to access feature of the program. Be sure to use Help to access the manual pages. The manual is included in the install package and is loaded locally for quick access.

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

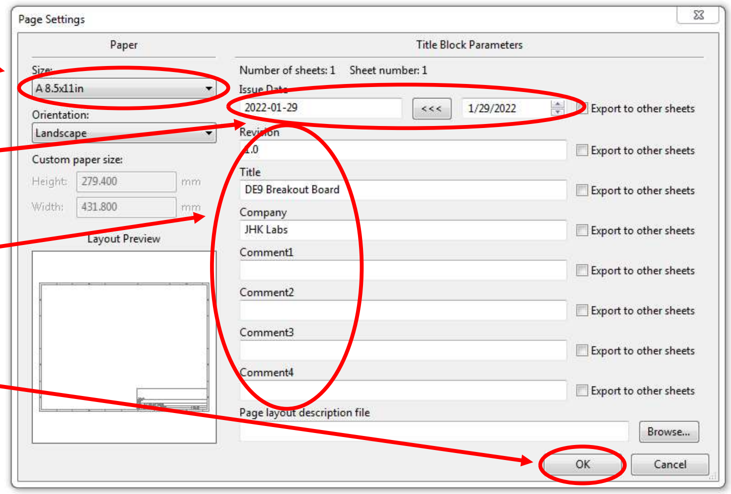
Schematic Layout Editor

Set Paper Size

Set Date

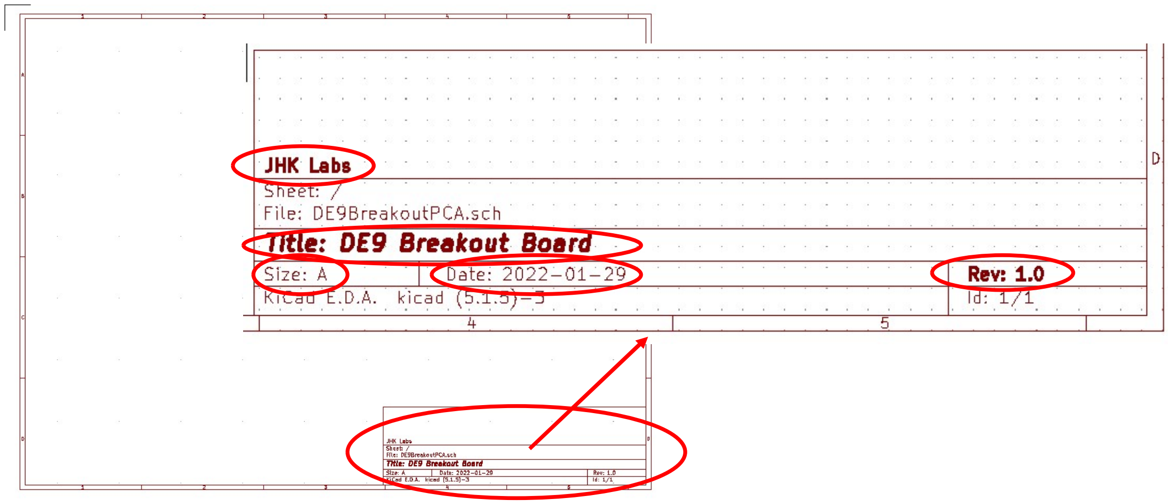
Set Title Block Info

Click OK



Explain: A,B,C,D,E are common. Use A for simple. Use B for large complex symbols like a microcontroller.
When done click OK.

Schematic Layout Editor



Sheet is A size, 8.5x11.
Title block contains your entries

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

The left toolbar sets modes. I use the defaults. Adjust to your liking as needed.

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

Grayed descriptions are for working with multiple sheets in a schematic. We’re not doing that today.

Schematic Layout Editor

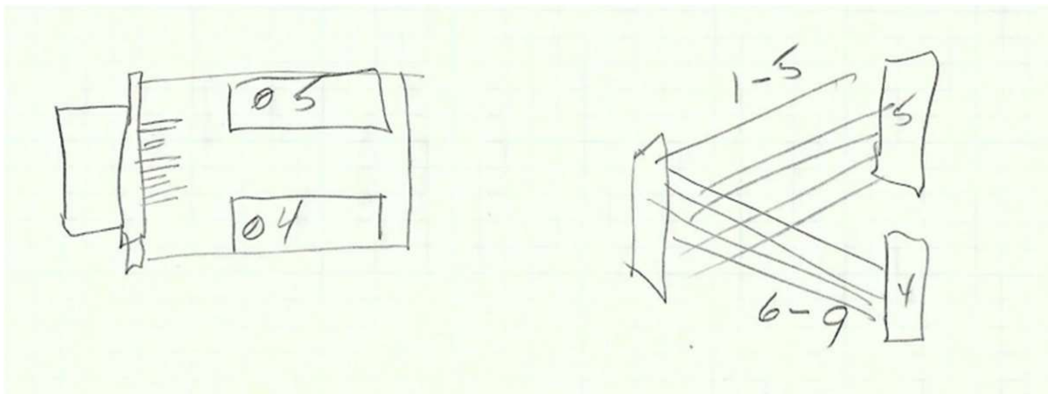
- Right Toolbar

  - 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

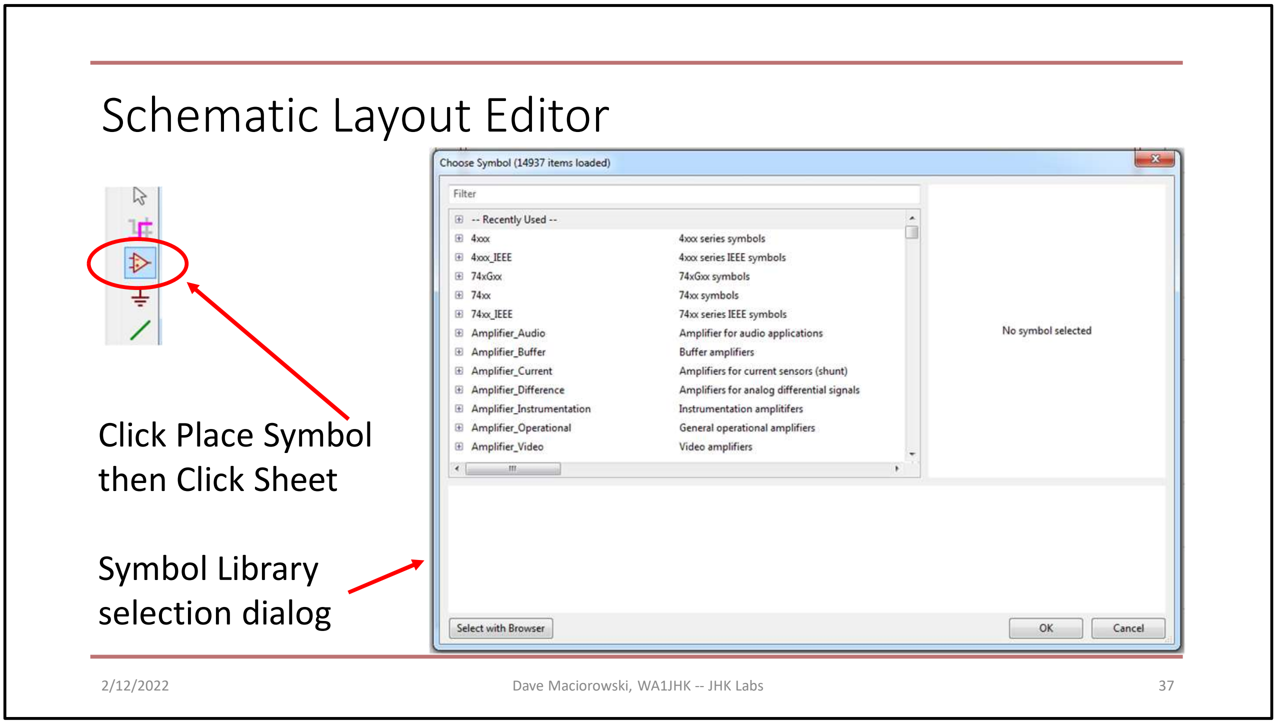


Here's the example project, again. The diagram on the right side is the rough schematic.
We need to put that into the Schematic Layout Editor.
We need to pick some parts.

Example Project – Parts

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 36

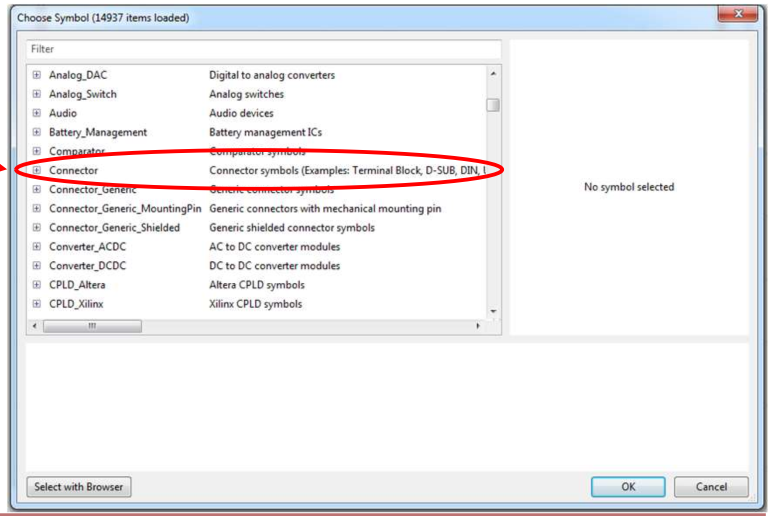
On the left are a couple examples of DE9M breakout boards. These use a PCB Mount connector.
Instead, we are going to use a DE9 male connector like you would solder on the end of a cable. We are going to use this connector as a straddlemount connector. Take a look at this middle picture. Our circuit board will fit nicely between the upper and lower row of connector pins.
On the right are the parts we're going to use.



Click Place Symbol from the Right Toolbar. The Choose Symbol dialog is displayed.

Schematic Layout Editor

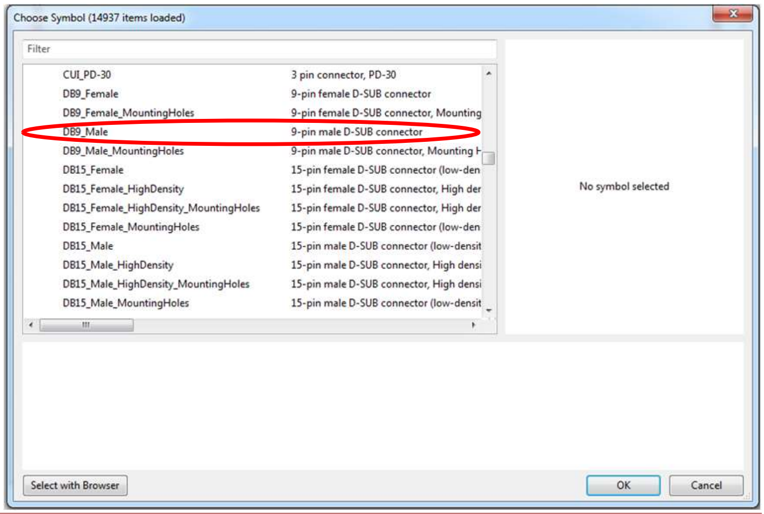
Scroll to Connector Category



As you scroll thru the list you'll see how many categories of symbols there are.
Open the Connector category,

Schematic Layout Editor

Scroll to DB9_Male Connector



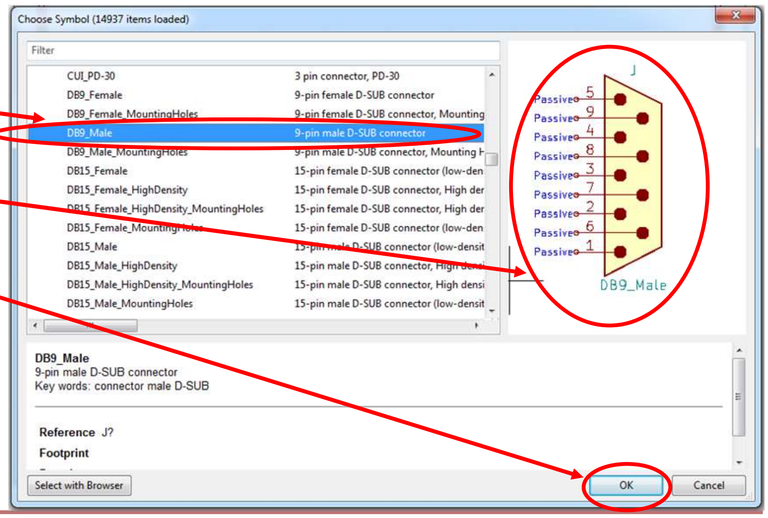
Scroll down to the DB9 connector. (A misnomer. It's actually a DE9.)
Click to see the symbol.

Schematic Layout Editor

Click Symbol Name

Review Symbol

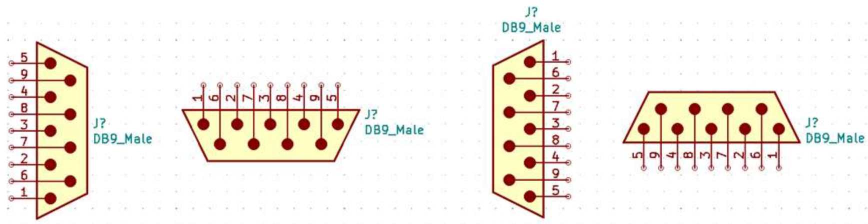
Click OK to Select



Just like the physical connector, pins 1 thru 5 are on the top, 6 thru 9 are on the bottom. Click OK to place.

Schematic Layout Editor

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



Once it is selected, you can rotate the symbol before placing it on the sheet. Press R for each 90-degree rotation.

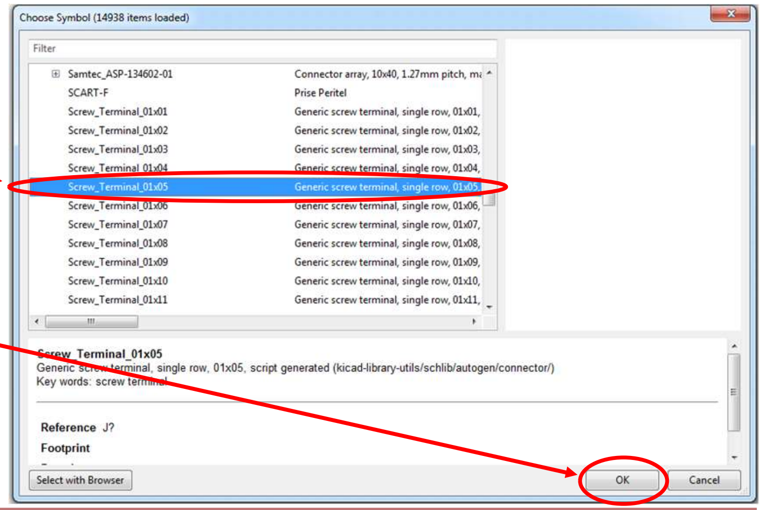
Schematic Layout Editor

Click Sheet

Scroll to Screw_Terminal_01x05

Click OK

Click to Place on Sheet



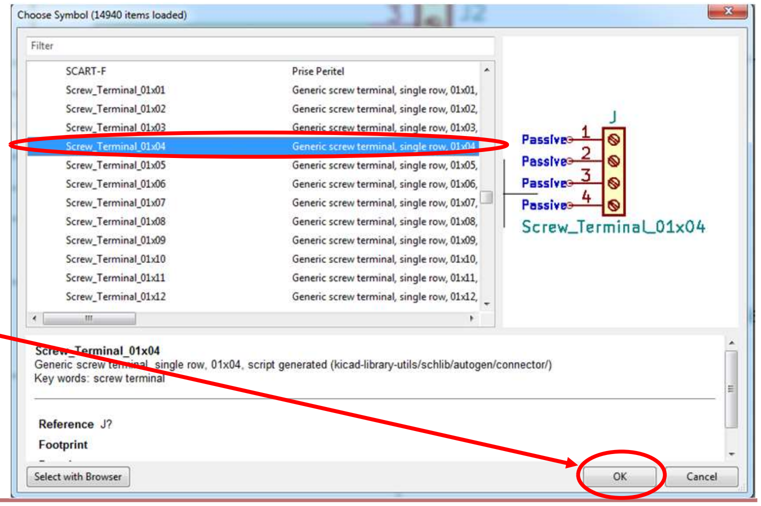
Schematic Layout Editor

Click Sheet

Scroll to Screw_Terminal_01x04

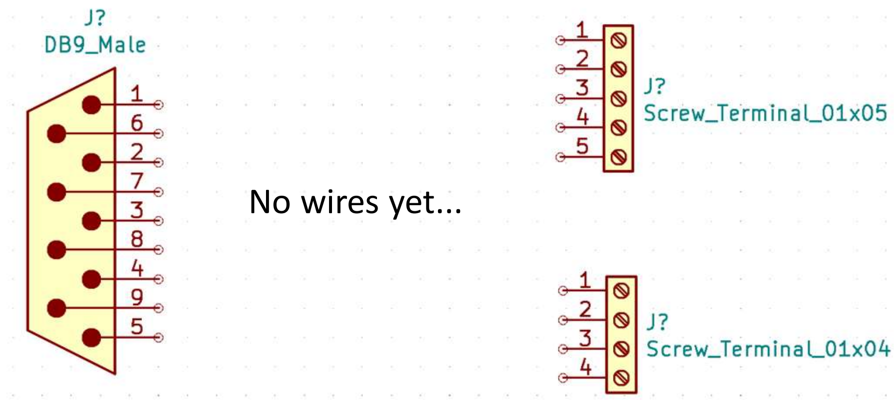
Click OK

Click to Place on Sheet



Schematic Layout Editor

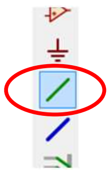
Schematic So Far...



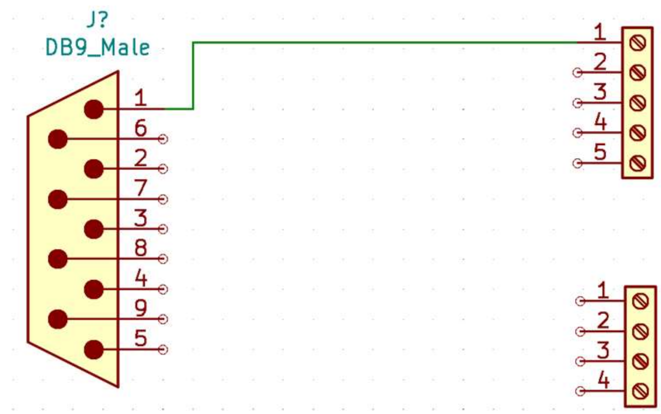
Here’s the schematic so far. Now to add wires.

Schematic Layout Editor

Click Place Wire



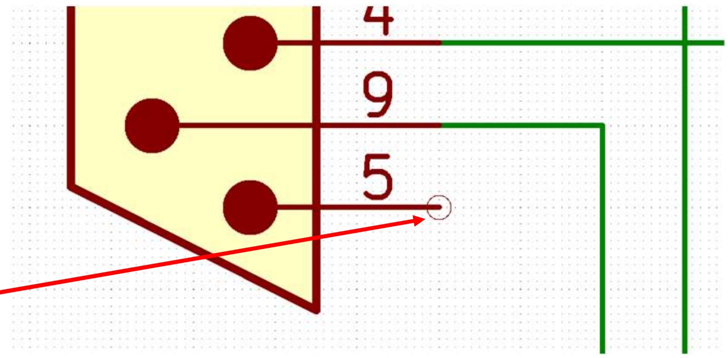
Click Start “Bubble”
Click Corner
Click End “Bubble”
On Each Wire...



To place a wire, we click the Place Wire tool on the right toolbar. Click the “bubble” on pin 1 and run it over to pin 1 on the 5-pin terminal strip and click the bubble there. First wire done. Do the same thing for the other 8 wires.

Schematic Layout Editor

Zoom In for Detail
on “Bubbles”

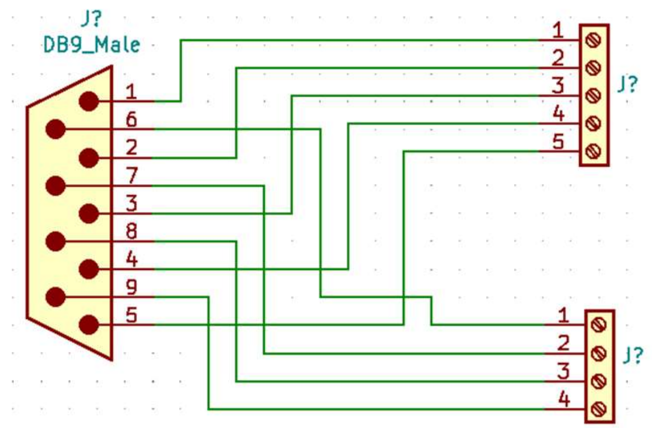


Click in the Middle
of the “Bubble”

Here’s an enlarged view of the connection point to pin 5 of the DE9 connector. You must click in the middle of the “bubble” to make the connection.
ProTip: Zoom In to click in the bubble, Zoom Out to route across the schematic page, Zoom back In to click at the destination.

Schematic Layout Editor

Schematic Done!

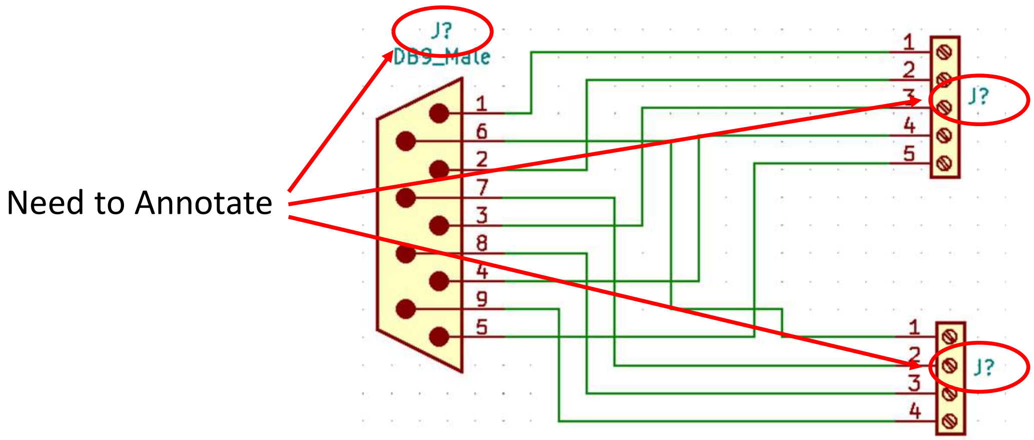


Schematic is all wired. We wired DB9 pin 1 to first terminal strip pin 1, then thru 5. This is the top row of pins on the DE9. We do the same for 6 thru 9 to the second terminal strip.

KiCad – Work Flow

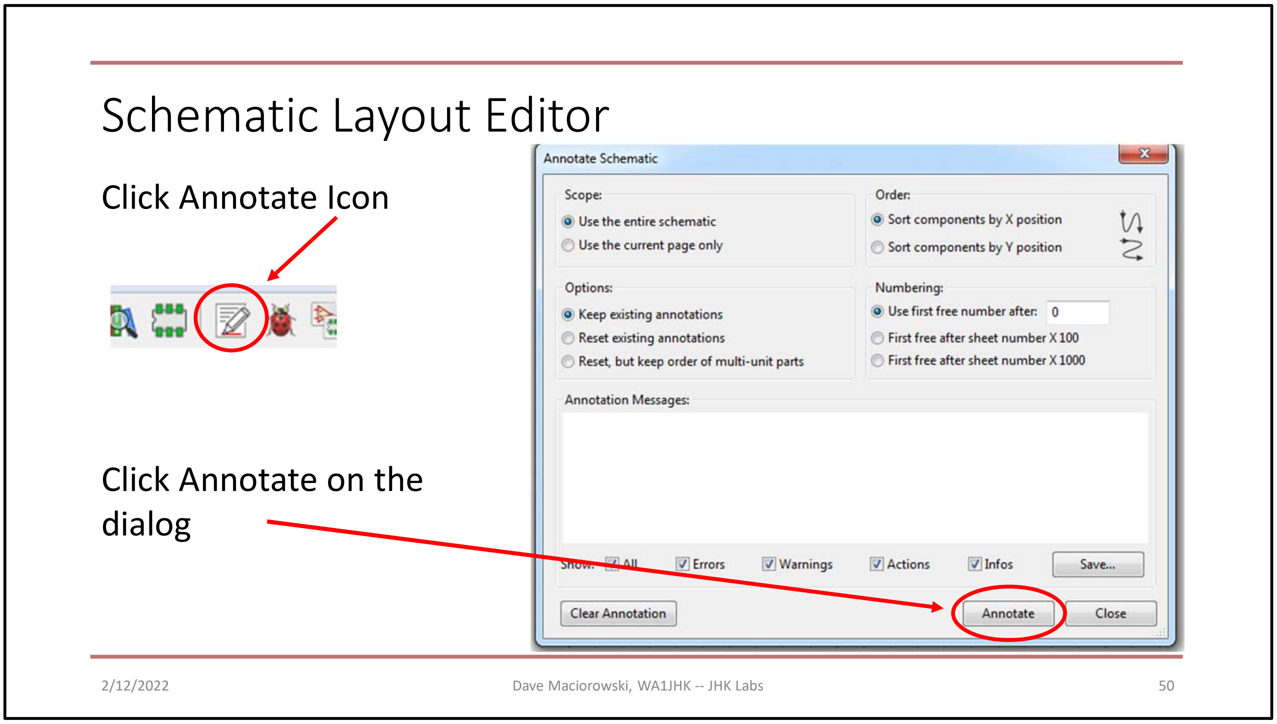
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate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



2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 49

See the J question mark on each connector? Those are the reference designators and need to be numbered. That's called Annotation.

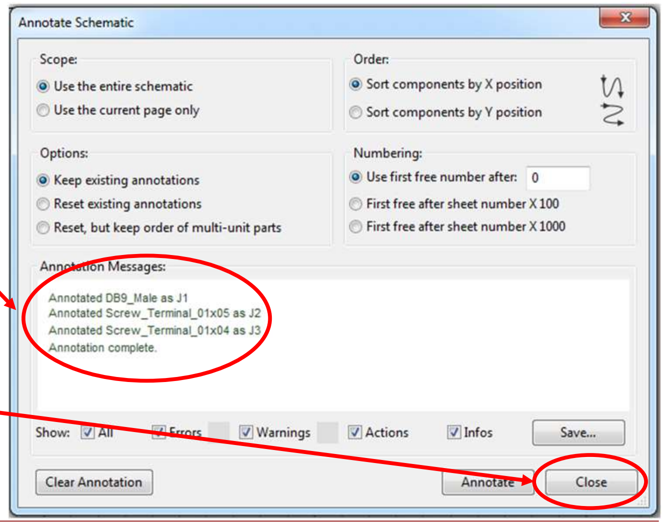


On the top toolbar, click Annotate. That brings up the Annotate Schematic dialog. For most applications, I use the defaults.
Click the Annotate button.

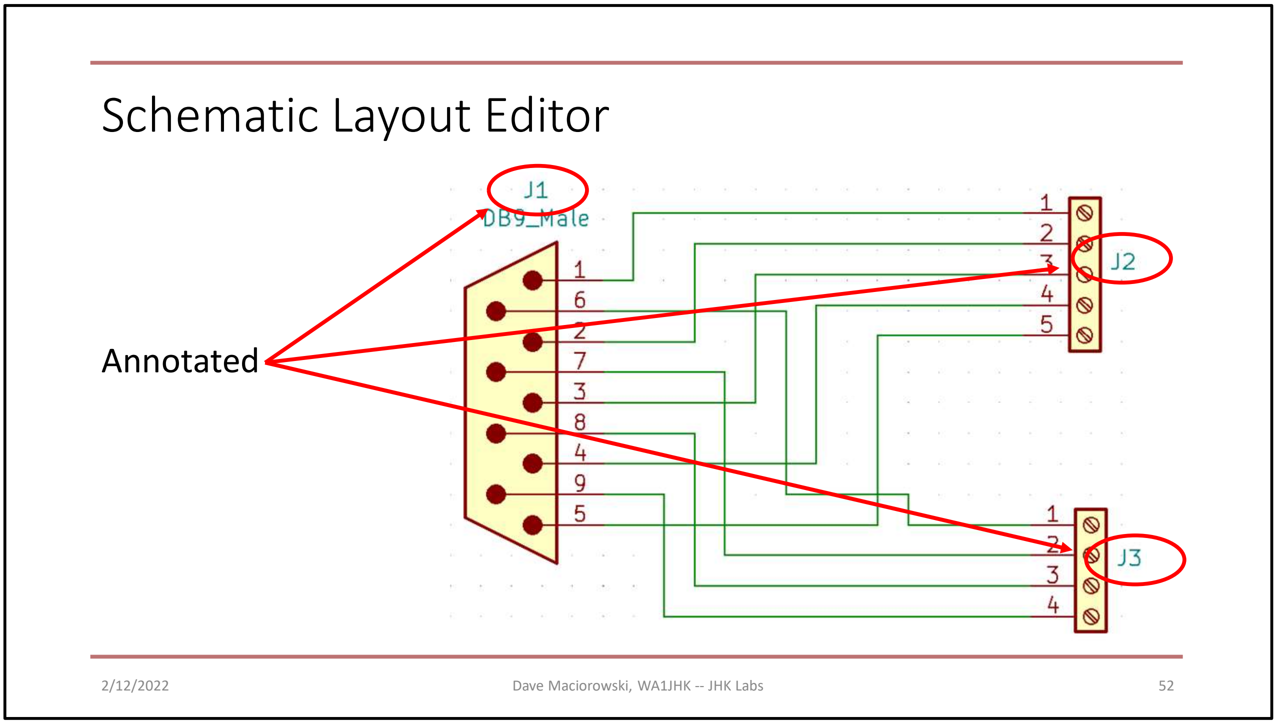
Schematic Layout Editor

Annotation Results

Click Close



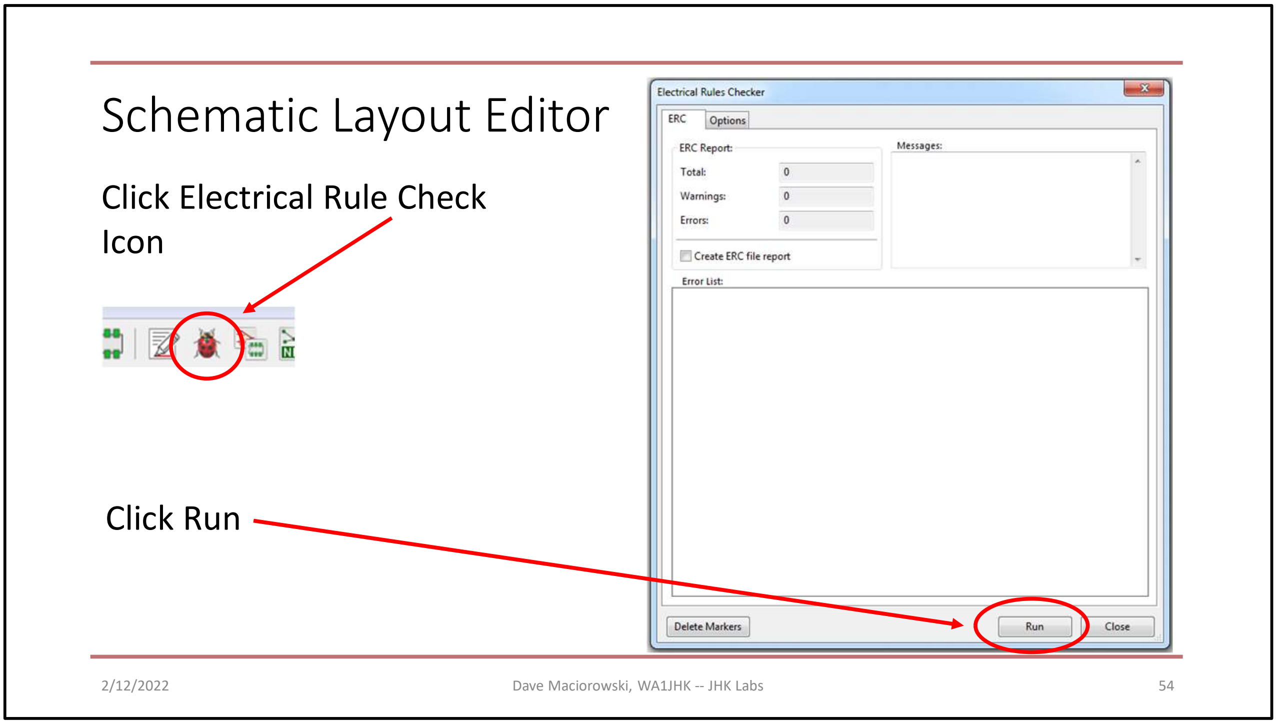
As Annotation processes the components on the schematic, it lists a log of what it did, shown here. Click Close when done.



Here’s the schematic after annotation.

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



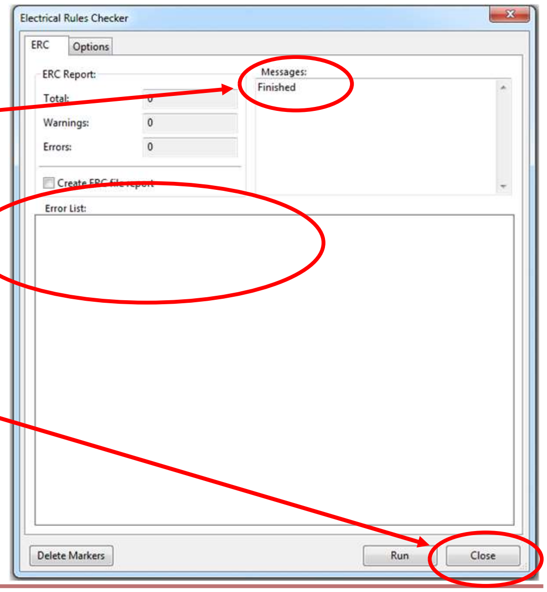
Click the Electrical Rule Check Icon. On this dialog, click Run.

Schematic Layout Editor

Finished, No Errors

Check for Errors

Click Close

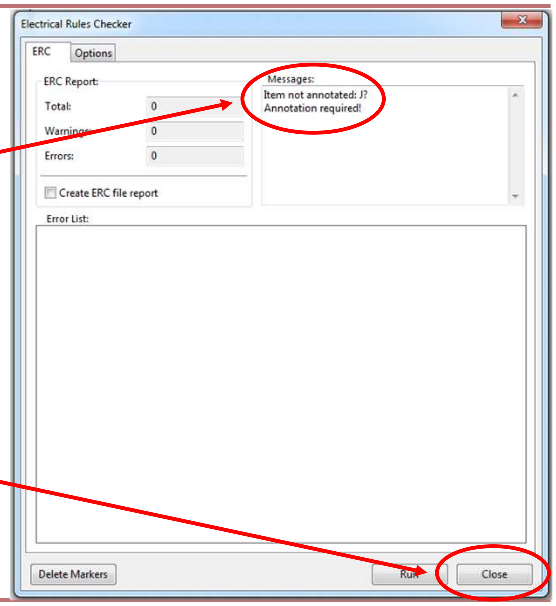


Since we connected all the pins on the connectors, there are no errors. Always look carefully at the Messages and Error List.

Schematic Layout Editor

Example of Running the Electrical Rule Check Before Annotation

Click Close



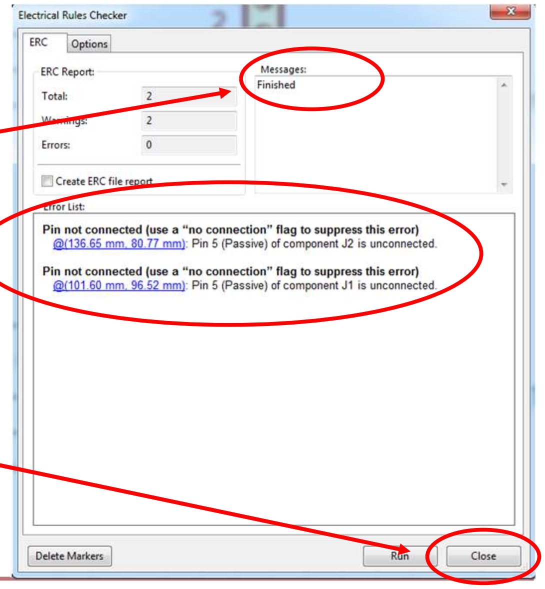
As an example, here's the report if the schematic is not annotated. Just go back to annotation, then run the Annotation.

Schematic Layout Editor

Example of Running the Electrical Rule Check with a Missing Connection

Check for Errors

Click Close

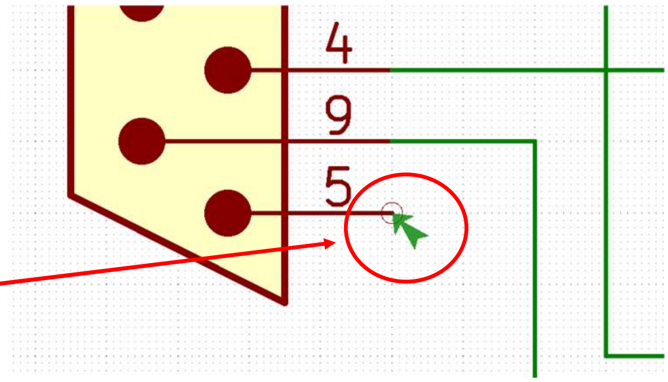


2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 57

As an example, here's the report if there's unconnected pins. Check for Errors. For these types of errors, the tool adds Markers on the schematic to help locate the issue.

Schematic Layout Editor

Green Marker
Arrows Point to Rule
Check Issues



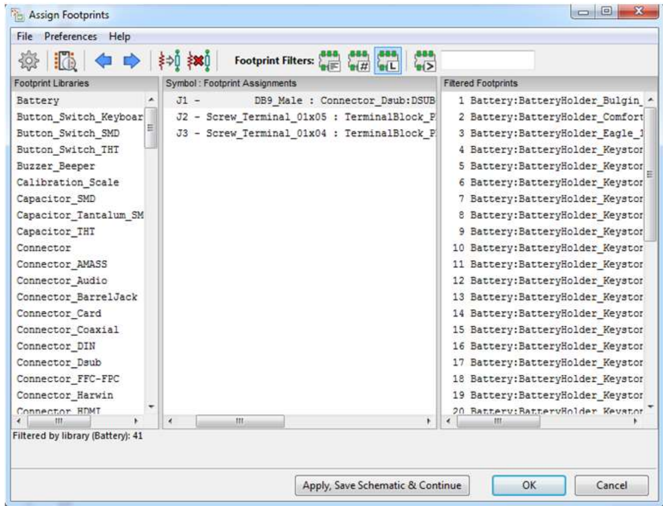
Green Marker Arrows point to issues found during the Electrical Rule Check. You need to resolve each of these issues. If the pin is supposed to be unconnected, there's a symbol in the right toolbar for that.

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



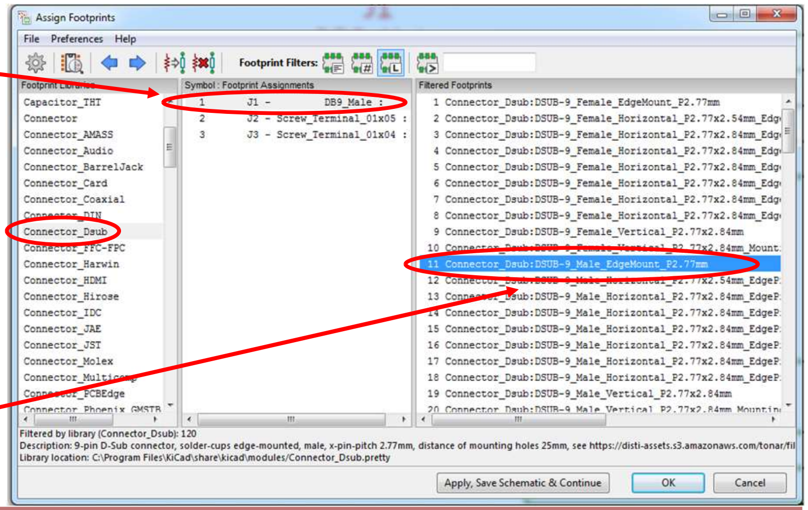
Click the Assign Footprints icon. We need to work down the list in the center and assign footprints for each symbol.
Tip: Column widths in this dialog are all adjustable. Spread it out as wide as your screen to be able to read everything.

Schematic Layout Editor

Click J1

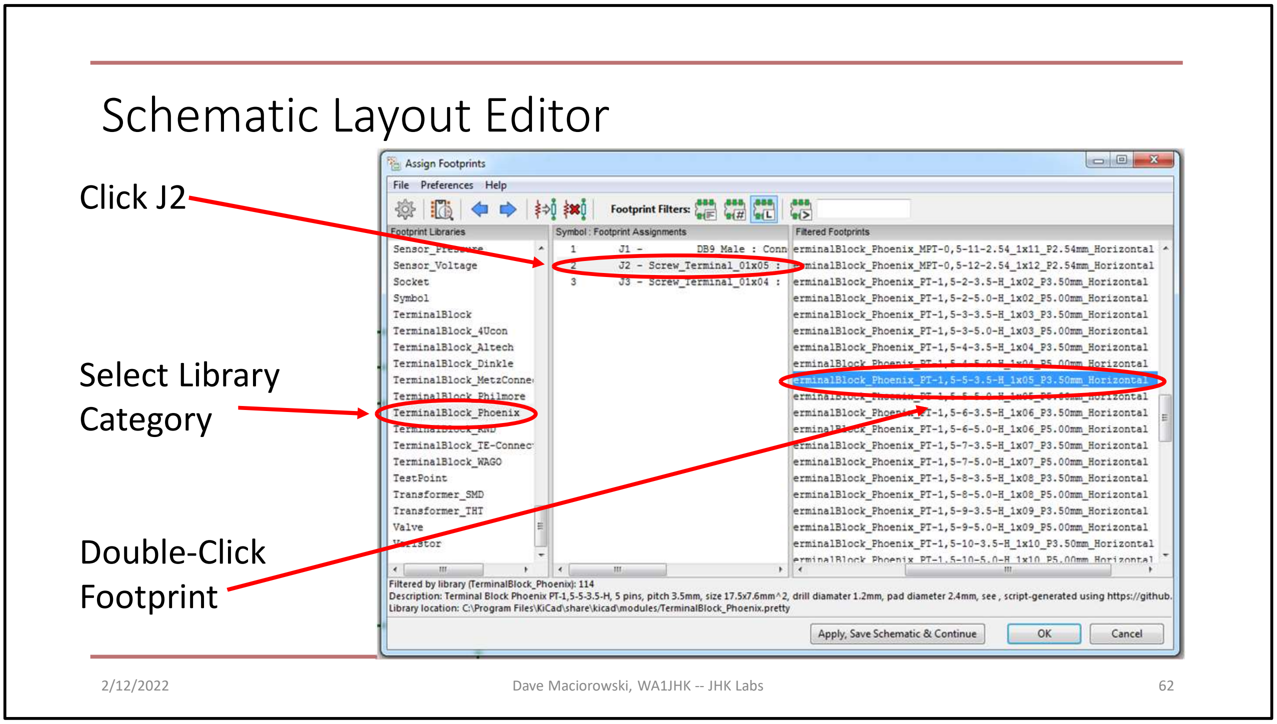
Select Library Category

Double-Click Footprint



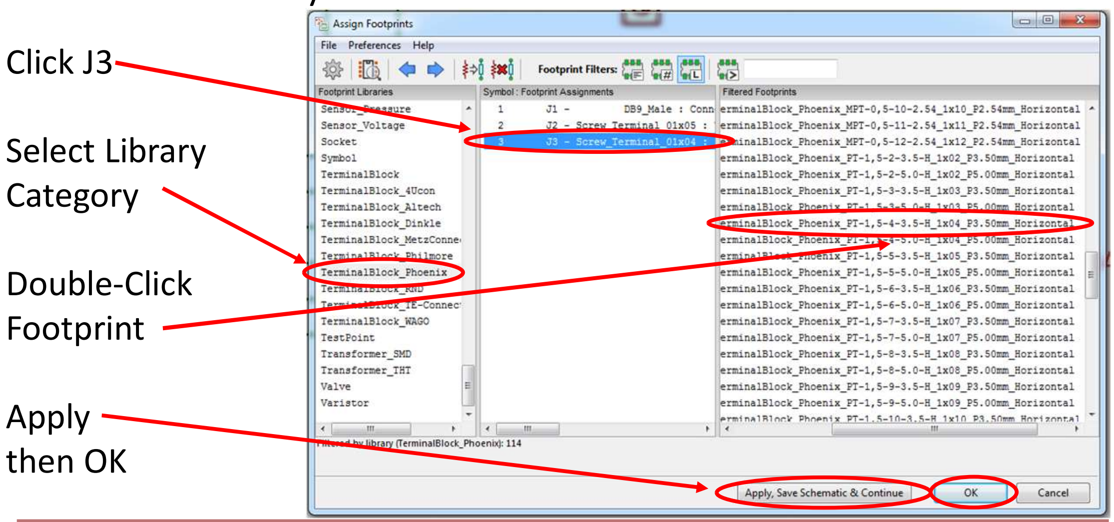
2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 61

Select the first component that requires a footprint.
Select which library the footprint should be stored in.
Double-click the footprint for your design.
Pro Tip: Use the Footprint Browser ahead of time to find what you need for your design.



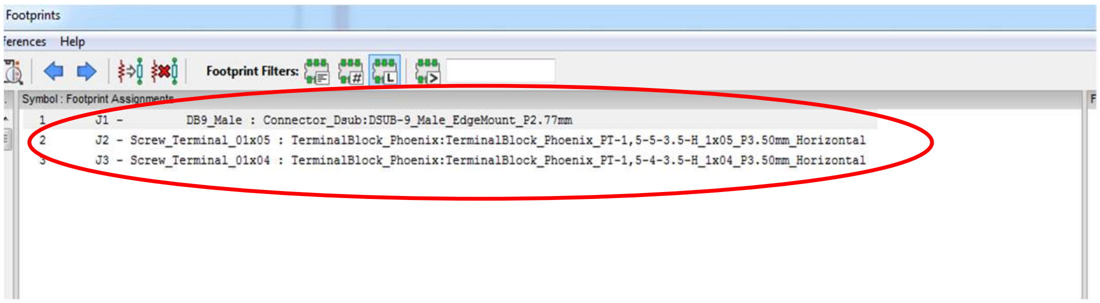
Select the second component that requires a footprint.
Select which library the footprint should be stored in. I happen to know that Phoenix has the terminal strip footprint we need for these 3.50 mm pitch terminal strips.
Double-click the footprint for your design.
Pro Tip: Use the Footprint Browser ahead of time to find what you need for your design.

Schematic Layout Editor



Select the last component that requires a footprint.
Select which library the footprint should be stored in, another terminal strip.
Double-click the footprint for your design.
Click Apply to your Schematic.
Click OK.

Schematic Layout Editor

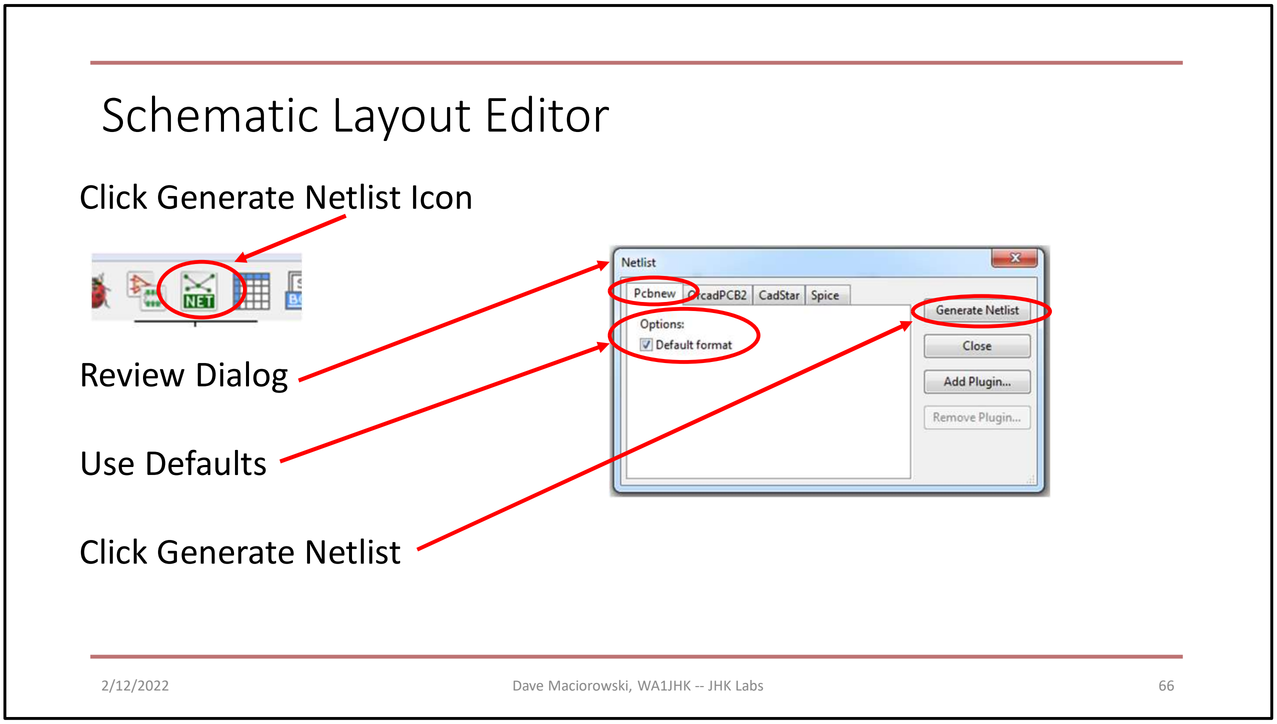


I widened the window to see the results.
Here are all the footprint assignments.

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

The last step in the Schematic is to Generate the Netlist

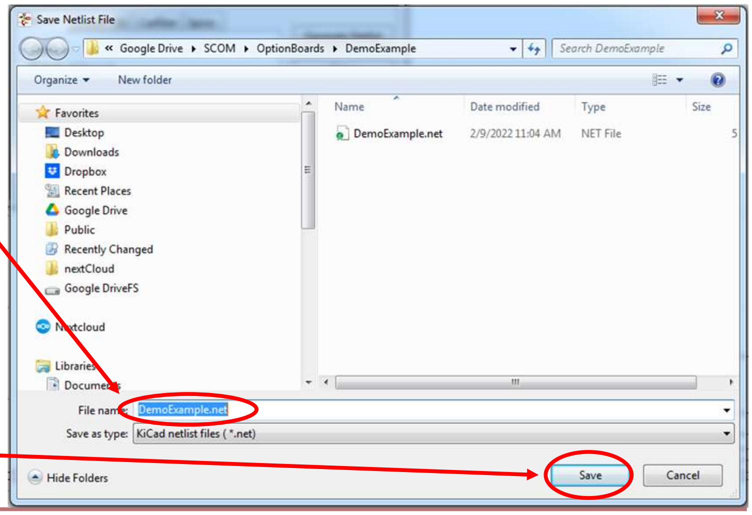


Use the defaults.
Click Generate netlist.

Schematic Layout Editor

Use Default

Click Save



Use the default filename.
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

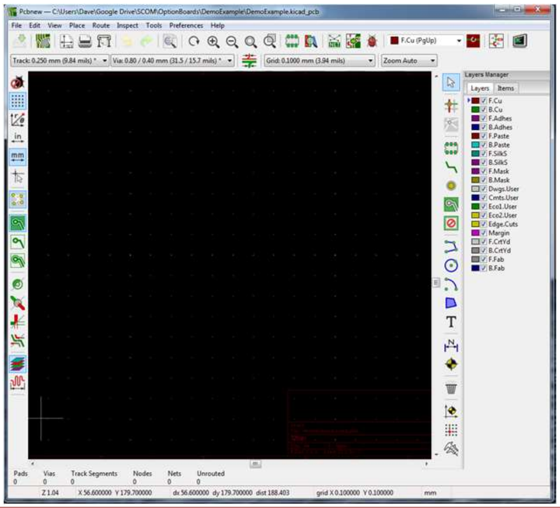
That completes the schematic.
We're now ready to lay out the circuit board.

Schematic Layout Editor

Click the
PCB Layout Editor

On the Top Toolbar, click the PCB Layout Editor icon.
This icon works on the Schematic screen or the Main screen.

PCB Layout Editor



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

This is the PCB Layout Editor main screen.
Yes, there's more icons in the toolbar.
[Summary above]

PCB Layout Editor

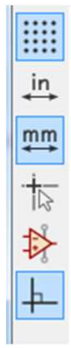
- Top Toolbar
  - Identical to the Schematic Layout Editor Toolbar except for a few Icons we aren't using for today's example.

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 71

Here's the toolbar. It's the same as the Schematic Layout Editor with a few new icons we're not using today.

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

I use the defaults. Adjust to your liking.

PCB Layout Editor

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

There's lots more settings that can be useful in more complex designs. See the tutorials and help file.

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

I use the defaults. Adjust to your liking.

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

I use the defaults. Adjust to your liking.

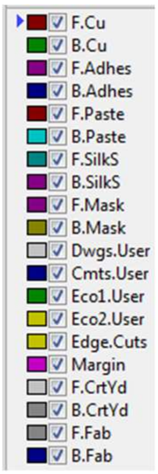
PCB Layout Editor

Right Toolbar

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

I use the defaults. Adjust to your liking.

PCB Layout Editor



Layer Manager

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

The Layer Manager is an important tool that we will be using today. These control the layer displayed in the drawing area AND the blue arrow selects the layer being drawn on.
The checkmarks enable that layer of the design to be viewed in the drawing area of the PCB Layout Editor.
The blue arrow selects the layer that will be affected when a modification to the design is made.
More about the Layer Manager when we load the Netlist.

PCB Layout Editor

Auxiliary Toolbar Mode Selections

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

With these tools, you can modify the width of the Track (trace) and the size of any Via that you place.
The Grid that is displayed and the grid that the routes and components are how accurately you can place footprints and traces.
We’re going to use the defaults for our design.

KiCad – Work Flow

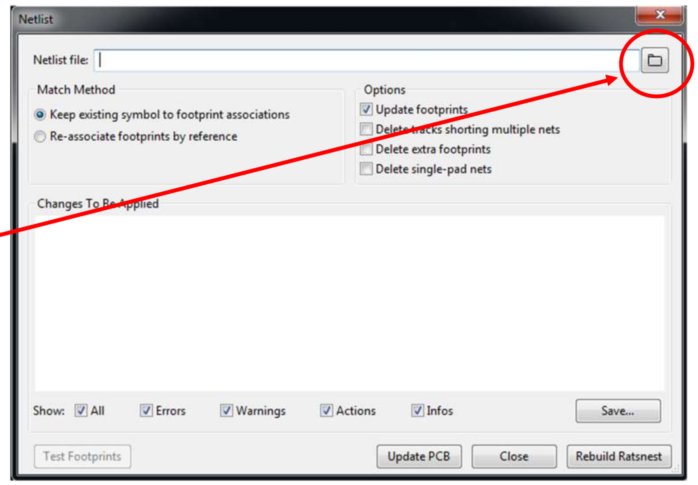
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate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

That’s the overview of the tools. Let’s get started on the circuit board.
We start by importing the NetList.

PCB Layout Editor

Click Load Netlist

Select File

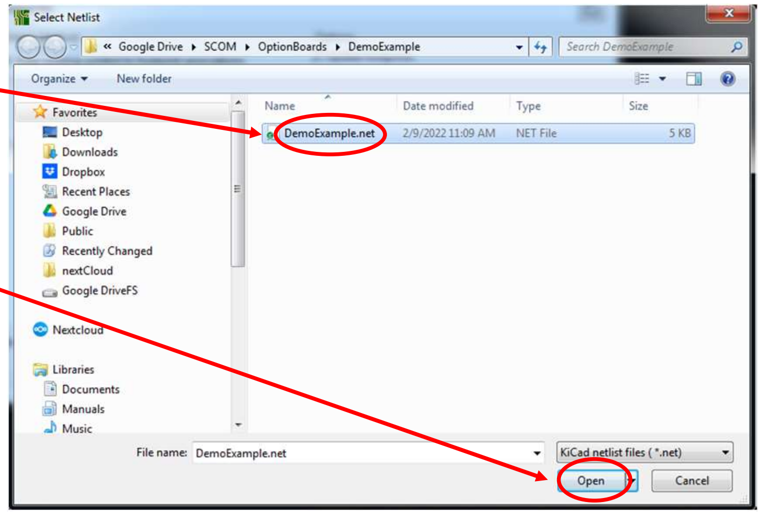


Click the Load Netlist icon on the Top Toolbar.
The Dialog is displayed. Click the Folder icon.

PCB Layout Editor

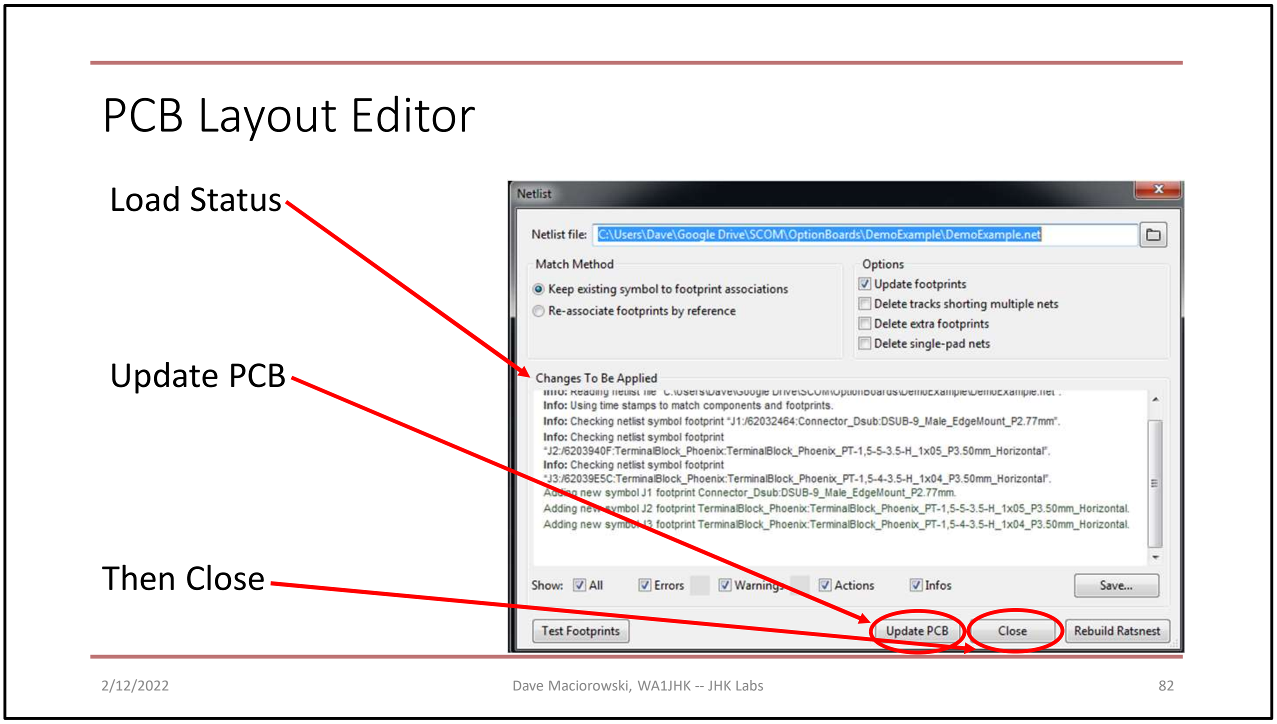
Select File

Click Open



2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 81

In the Open File dialog, select the Netlist File from the file list – that's the one with the .net extension, listed by default. This is the file you generated in the Schematic Editor.
Click Open



The Netlist is loaded. The status generated during the load is displayed on the screen. Review it carefully!!!

You need to resolve any errors you see. This might be a missing footprint. You'll need to go back to the schematic editor, fix the footprint, regenerate the netlist, then come back here and reload the netlist. Make sure all errors are resolved before continuing.

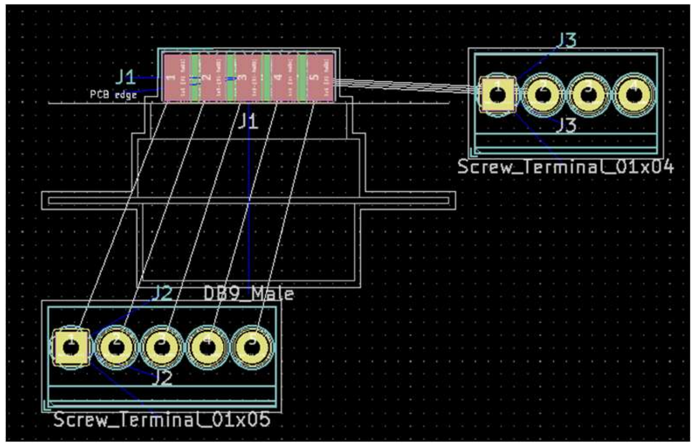
Click Update PCB, then Close.

PCB Layout Editor

Click to place the Rats in the Drawing Area

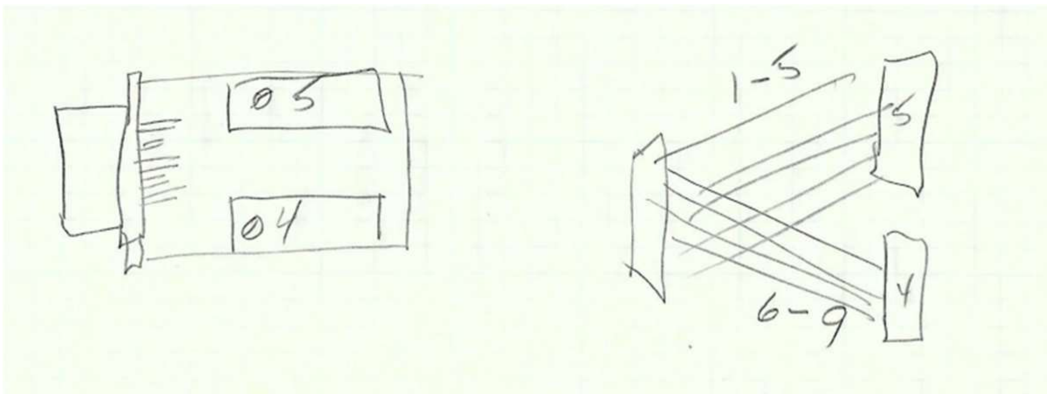
Click, press M and Move each Component to Position

R to Rotate



The Netlist contains a list of the components and interconnections, but it doesn't know where you want them placed on the PCB. It's called a Rats Nest or Rats for short.
On a complex board, the Rats can be very entertaining to untangle. This is why it's important to have some idea of how the board is going to be laid out.
Generally, the connectors and terminal strips define the placement.

Example Project – Hand Sketch



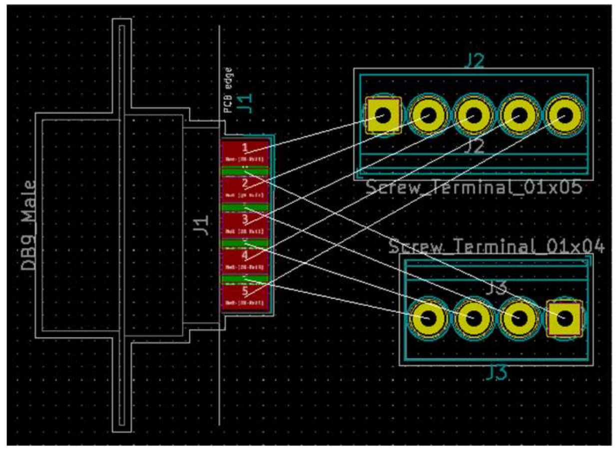
Remember the hand sketch. On the right is the wiring of the DE9 connector and the terminal strips that we took care of in the Schematic.
On the left is the layout we had in mind. Let’s move the components in the Rats to look like this.

PCB Layout Editor

The View is Always from the Top of the Board

Components placed

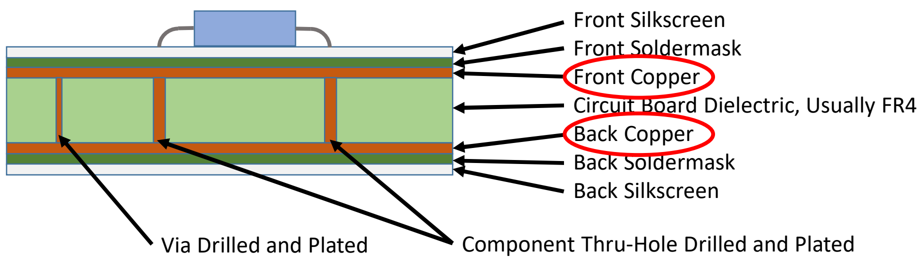
The white lines are the Air Wires. These are the Traces to be Routed.



The components have been rotated and placed about where we need them. It works like this. Click the component, press M for Move, drag to the location. Press R to rotate.
In this view, we haven't drawn the edge of the board yet. Take your time with placement and get an idea of how the board will look. You can always move them again later.

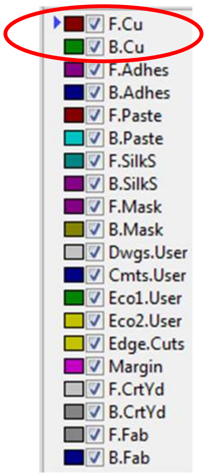
Anatomy of a PCB

- 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



Remember how the board is built. We’re focused right now on the Front Copper and the Back Copper.

PCB Layout Editor



Layer Manager

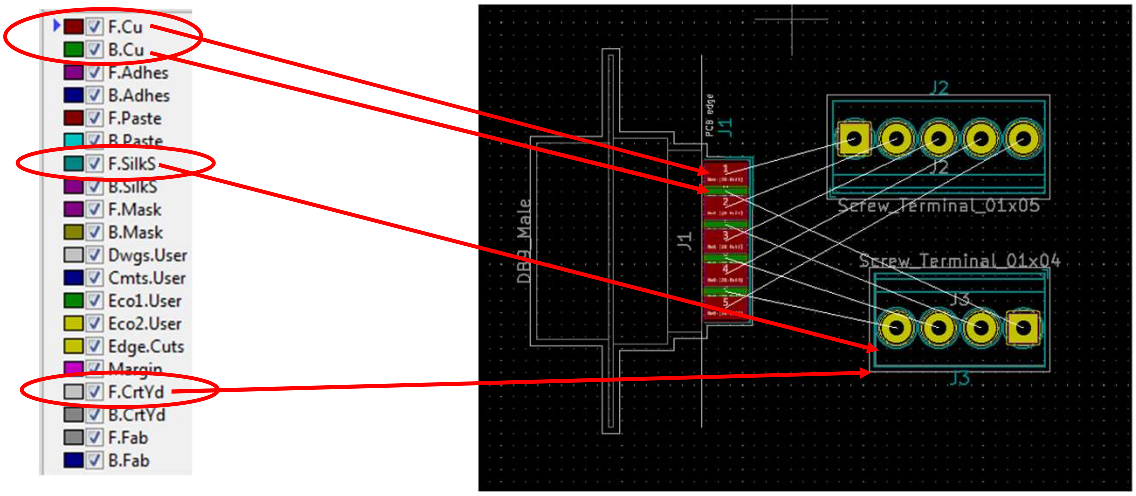
- Control Display of each Layer in the Drawing Area
- The Blue Arrow Defines Where Modifications Occur

This is where we use the Layer Manager. Notice the colors. The checkmarks enable that layer of the design to be viewed in the drawing area of the PCB Layout Editor in that color. The blue arrow selects the layer that will be affected when a modification to the design is made.

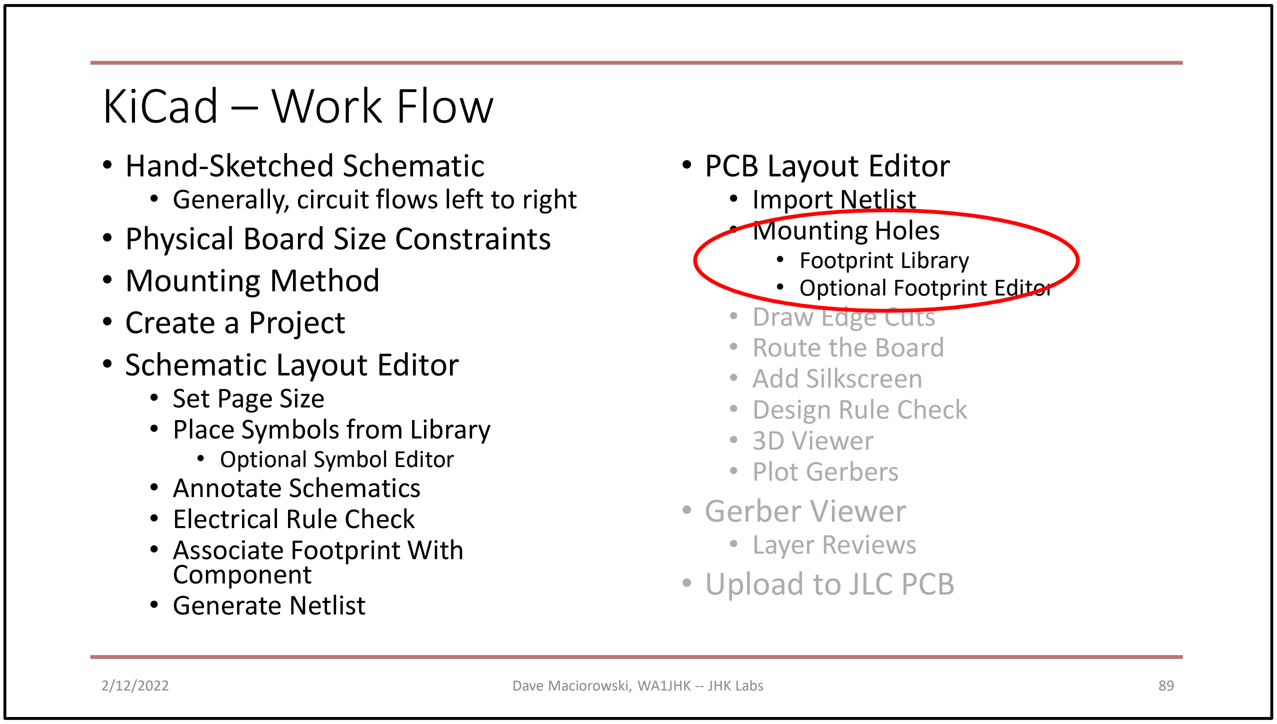
There's Front and Back Copper, Paste, SilkScreen, Mask, Edge Cuts and CourtYard. The other layers aren't important for simple designs.

We're going to focus on the Copper layers for now.

PCB Layout Editor



Here's our part footprints in the drawing area.
F.Cu is the Front Copper. B.Cu is the Back Copper. F.SilkS is the Front Silkscreen, the white printing on a circuit board.
F.CrtYd is the Front Courtyard. This is a physical boundary (in White) that the physical component could occupy. You don't want two devices in the same space. The Design Rule Check will check for no overlaps. We won't have any here because we're spread out so far.



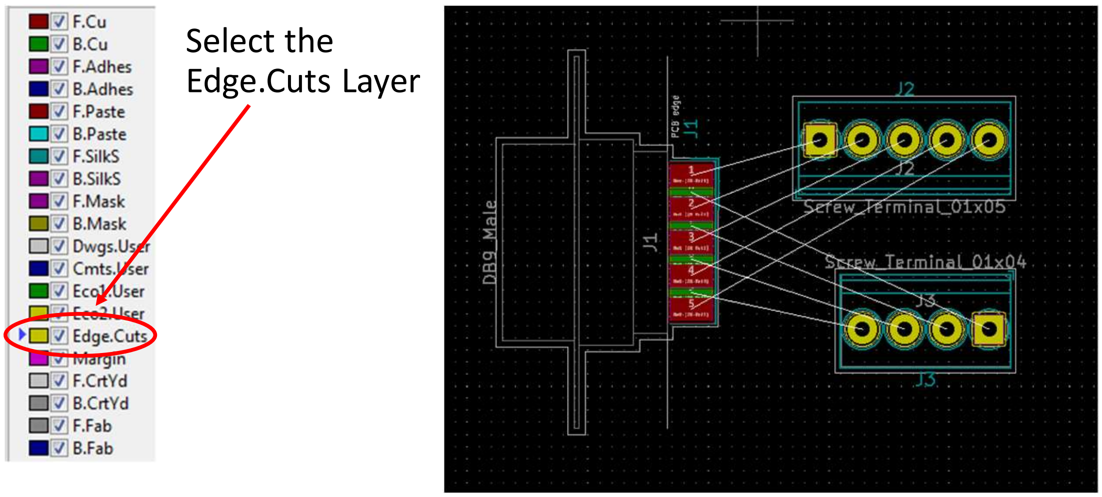
That completes the schematic.
We're now ready to lay out the circuit board.

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

That completes the schematic.
We're now ready to lay out the circuit board.

PCB Layout Editor



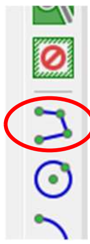
2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 91

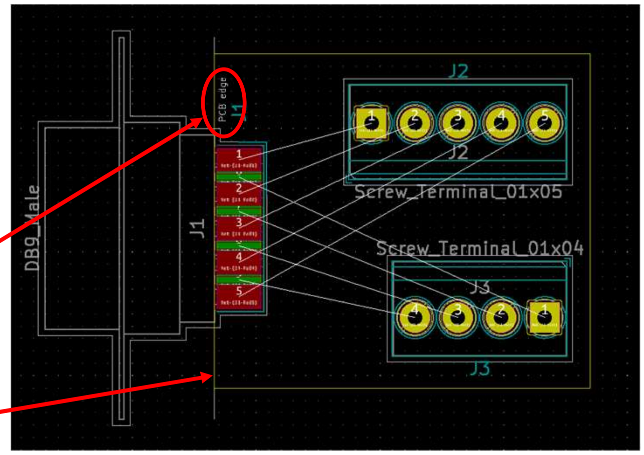
Click next to the Edge.Cuts line in the Layer Manager to move the Blue Arrow there. We are going to use the Add Graphics Line tool to draw the edge of the circuit board on that layer.

Note that the color of the layer. This is the color of the line we'll be drawing on that layer.

PCB Layout Editor

Click Add Graphics Lines





Note “PCB Edge”

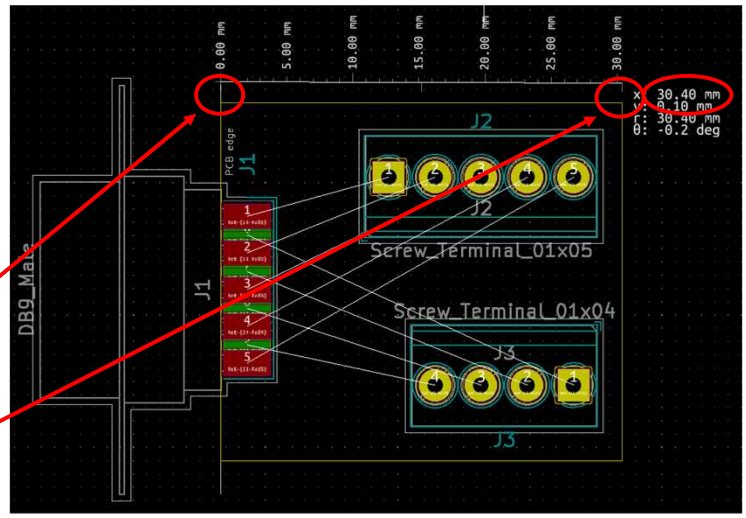
Draw a Complete Box
Around Components

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 92

I just drew the yellow box as the edge of the circuit board. The DE9 is designed to be hanging off the edge of the board.
Notice where it says “PCB Edge”. This should be where you draw the board edge for this footprint. I drew a line along the board edge then around the other components in a rectangle. Use the grid lines to guide you. Be sure to close the rectangle. Zoom in if needed to see the grid and corners.

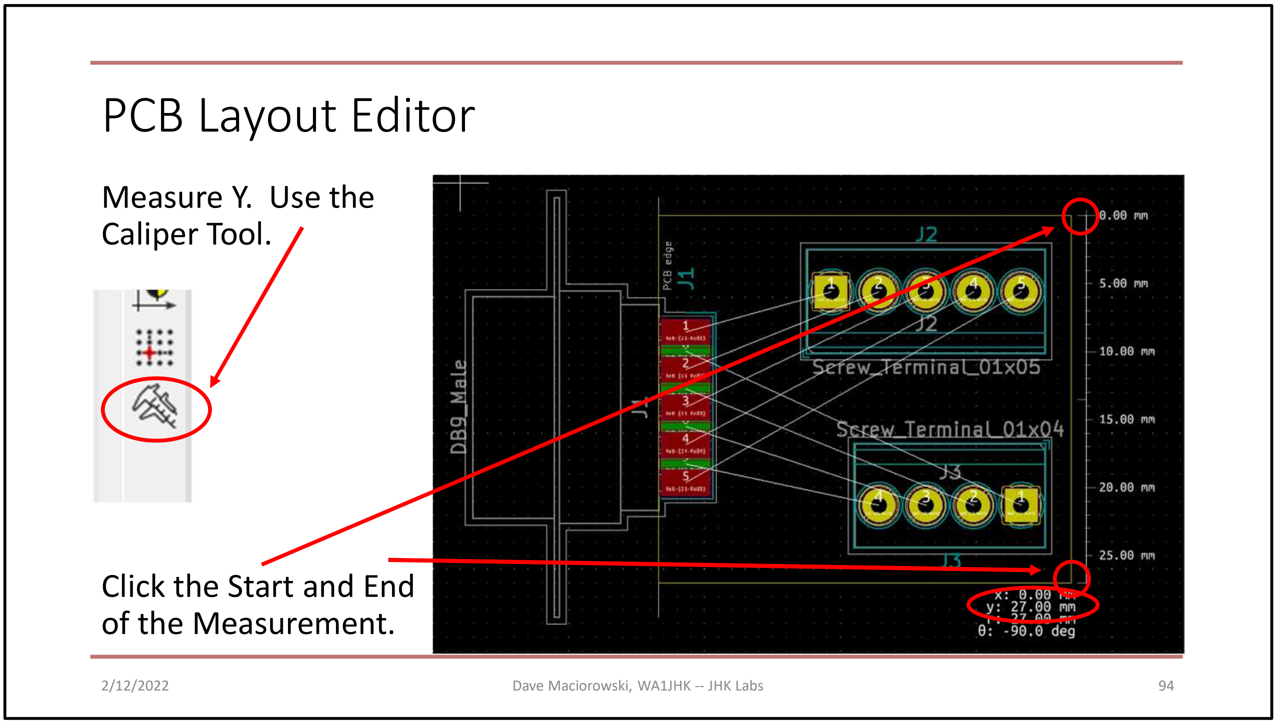
PCB Layout Editor

Measure X. Use the Caliper Tool.



Click the Start and End of the Measurement.

I like to measure the board size at this point. Click the Caliper Tool. Click the Start, Click the End. We see 30.40 mm in the X dimension.

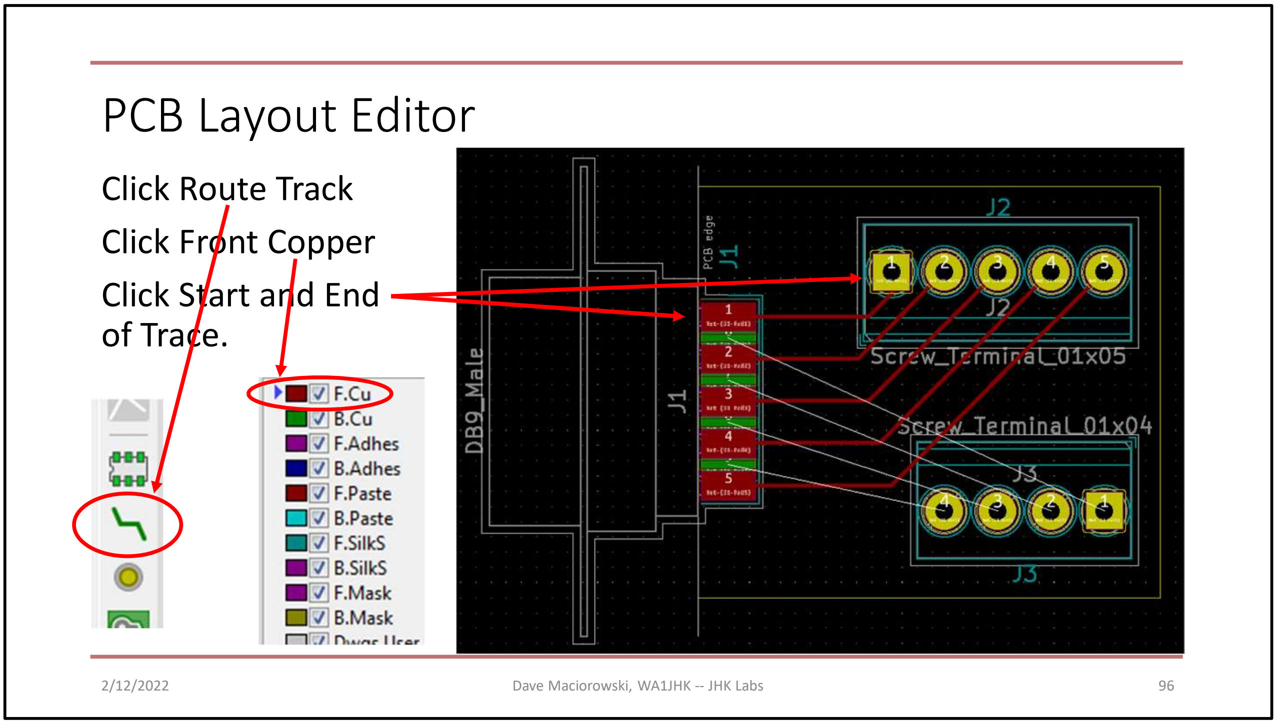


Do the same thing in the Y dimension. We see 27 mm in the y dimension.
At JLCPCB with their $2 special, 5 boards will cost $2 – 40 cents each – plus $20 for DHL to bring them here.
The quote for 25 boards is $6 – 24 cents each – plus $20 to ship them here.

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

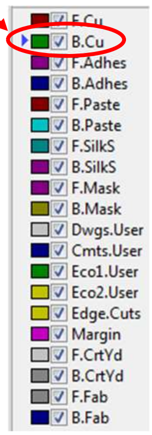
That completes the schematic.
We’re now ready to lay out the circuit board.

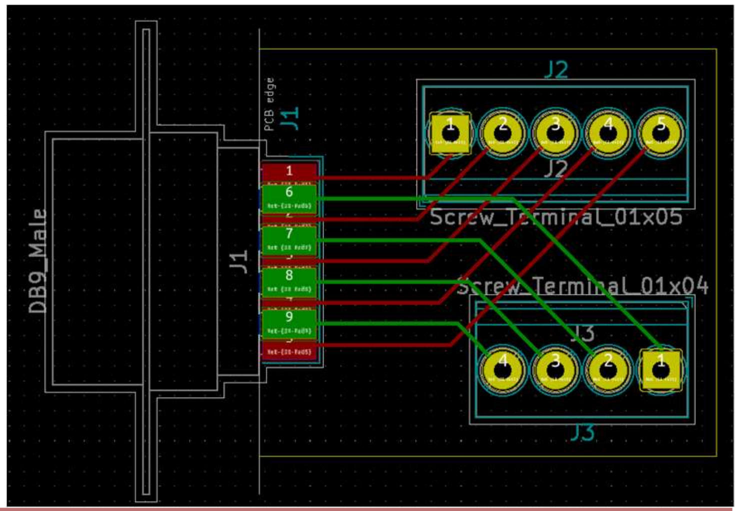


We’re ready to route the board.
Click one end of the Air Wire in the Rat that you want to create a Track for. Note how the Track is automatically routed by the tool. Click the other end to finish.
Click in the middle of the track to force the tool to make a turn and follow it your way.
This is the Front Copper side.
You can play around with the routes. You can erase the track and start over if you screw up or see a better way to route it. This is what “routing a board” is all about. It’s a puzzle.

PCB Layout Editor

Now the Back





Click to the left of the Green box for the Back Copper. Route the tracks the same way, this time in Green.

KiCad – Work Flow

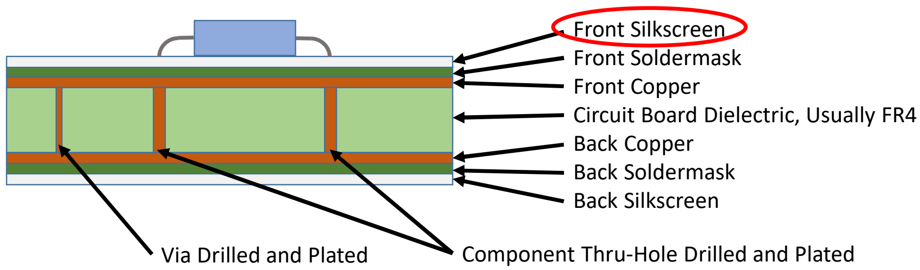
- Hand-Sketched Schematic
  - Generally, circuit flows left to right
- Physical Board Size Constraints
- Mounting Method
- Create a Project
- Schematic Layout Editor
  - Set Page Size
  - Place Symbols from Library
    - Optional Symbol Editor
  - Annotate Schematics
  - Electrical Rule Check
  - Associate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

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 98

That completes the schematic.
We're now ready to lay out the circuit board.

Anatomy of a PCB

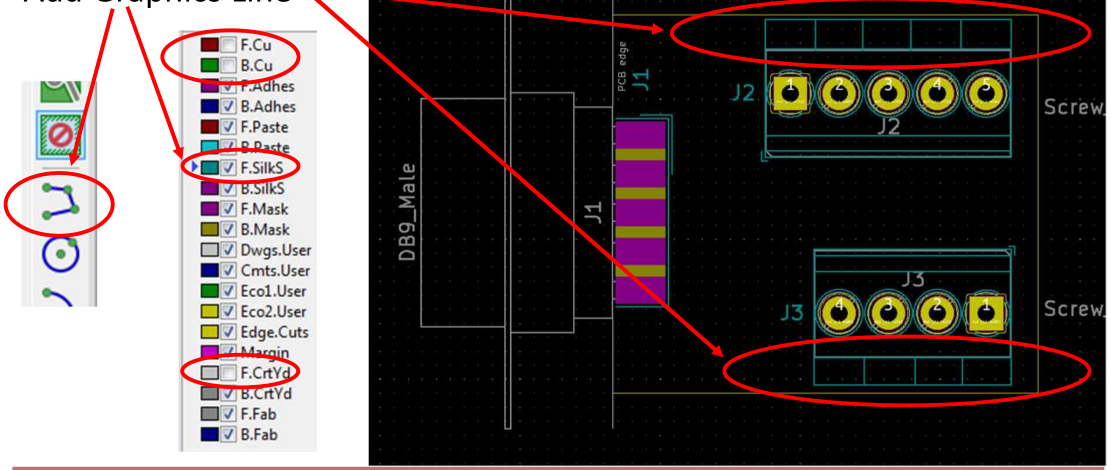
- Many Steps to Manufacturing a PCB
- “Gerber” Files Used in the Manufacturing Process
- KiCad Creates Gerber Files
- “Layers” Refers to the Number of Copper Layers, Always Even, Up to 30



Remember how the board is built. We’re focused right now on the Front Copper and the Back Copper.

PCB Layout Editor

Add Graphics Line

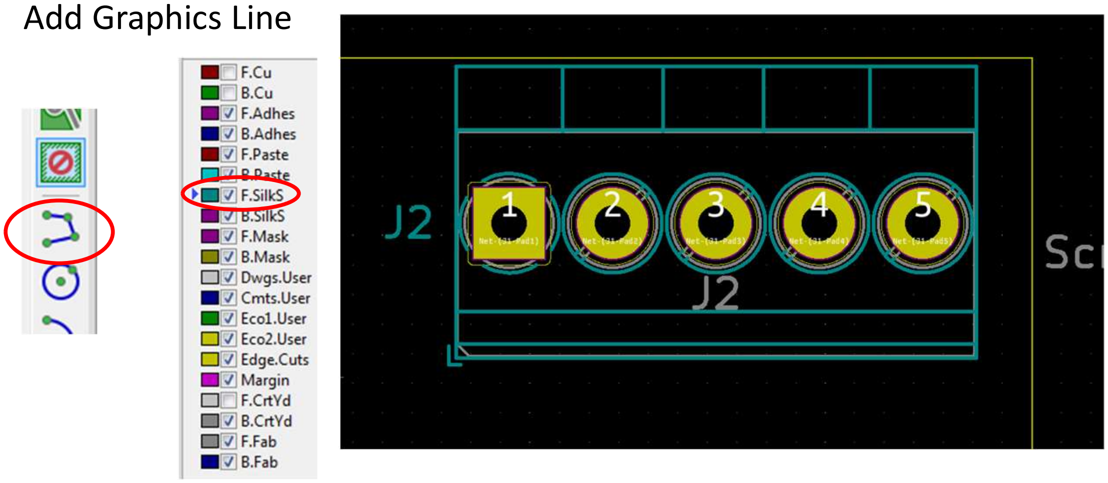


2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 100

I like to add some Silkcreen lines and labels to make it easier to use the finished board. I turned off (unchecked) the Front and Back Copper and the Front CourtYard to more clearly see the Silkscreen. Then I drew lines and boxes so I can number each pin.

PCB Layout Editor

Add Graphics Line

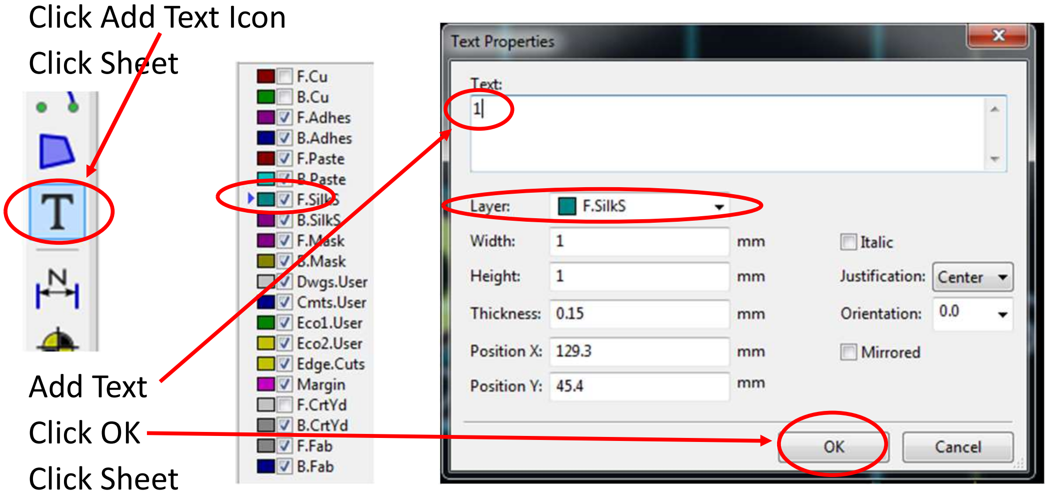


Here's a closeup of the lines I added. Be sure to Zoom in while drawing lines to make it easier on you and get a better result.

PCB Layout Editor

Click Add Text Icon

Click Sheet



Add Text

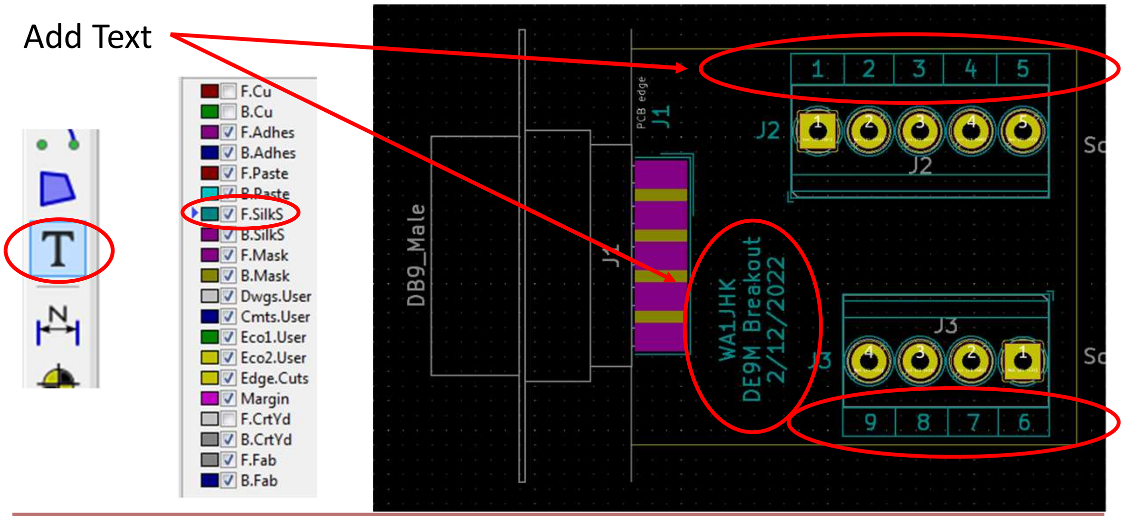
Click OK

Click Sheet

Now I want to add numbers for each of the terminals that represent the pin on the DE9 connector that is wired there.
Click the Add Text Icon. Click the Sheet. Fill in the dialog. Here we just add the text we want. Click OK, then position the text on the sheet.

PCB Layout Editor

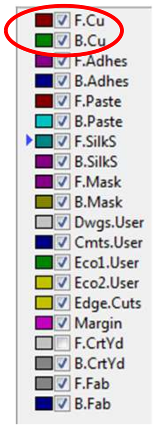
Add Text

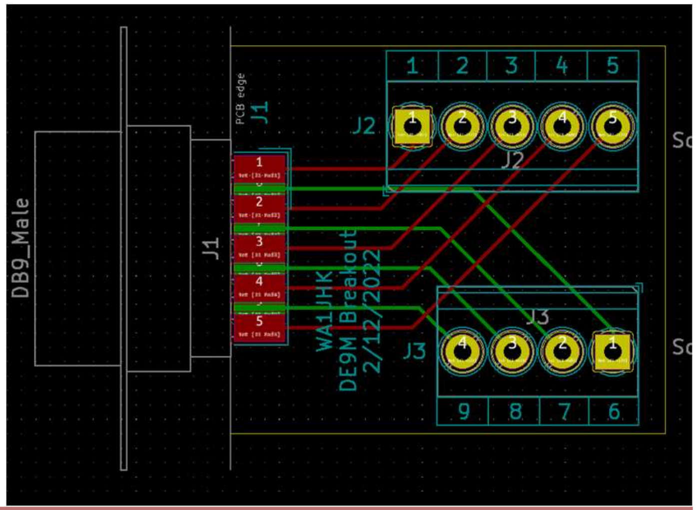


The terminal numbers are labeled. I added text identifying what this board is.

PCB Layout Editor

Copper Layers Back On





2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 104

Here's the finished board with the Copper Layers turned on. We have the Front and Back Copper layers, the Front Silkscreen, and the Edge Cuts that defines the board size.

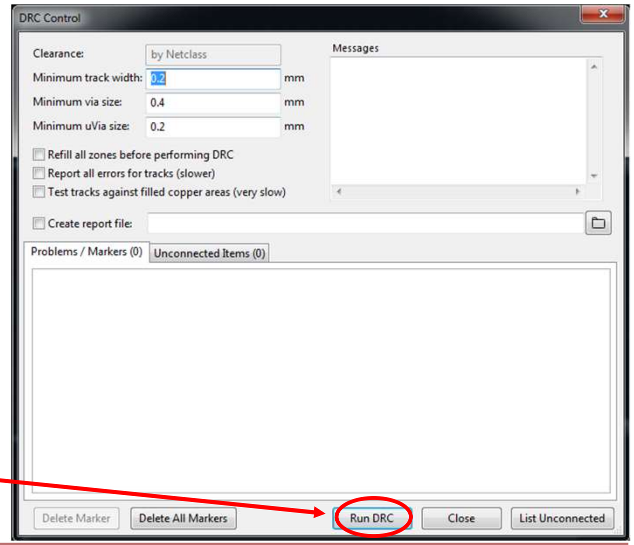
KiCad – Work Flow

- 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

That completes the schematic.
We're now ready to lay out the circuit board.

PCB Layout Editor

Click Design Rule Check



Click Run

Click the Design Rule Check. This dialog is presented. Click the Run DRC button.

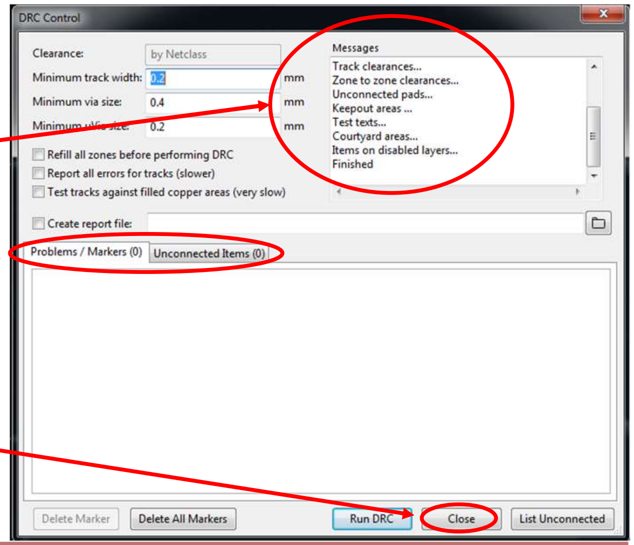
PCB Layout Editor

Review Messages

Look for “Finished”

Check Errors

Click Close



2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 107

Check the Messages showing DRC progress.
Check that the error counts are zero, especially Unconnected Items
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

That completes the schematic.
We're now ready to lay out the circuit board.

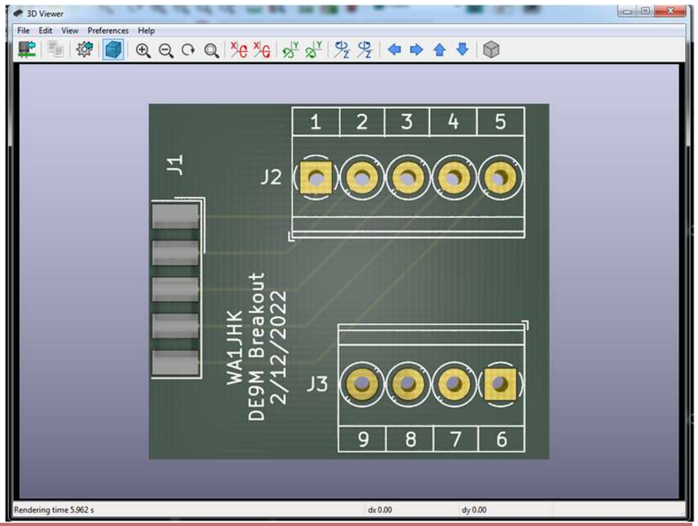
PCB Layout Editor

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.



2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 109

Press ALT-3 and be patient. The top of the board will be rendered. You can see what the board will look like when manufactured.
The Green is the Soldermask. The White is the Front Silkscreen. The Gray and Gold are the solder pads.

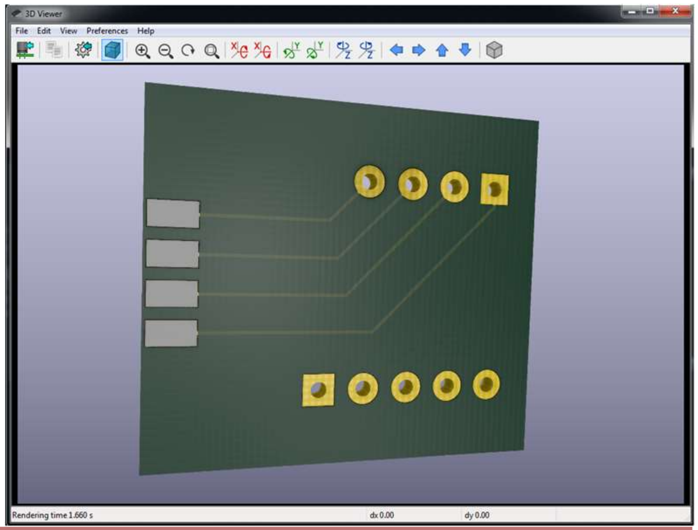
PCB Layout Editor

Press ALT-3

3D Viewer Displays in a new window.

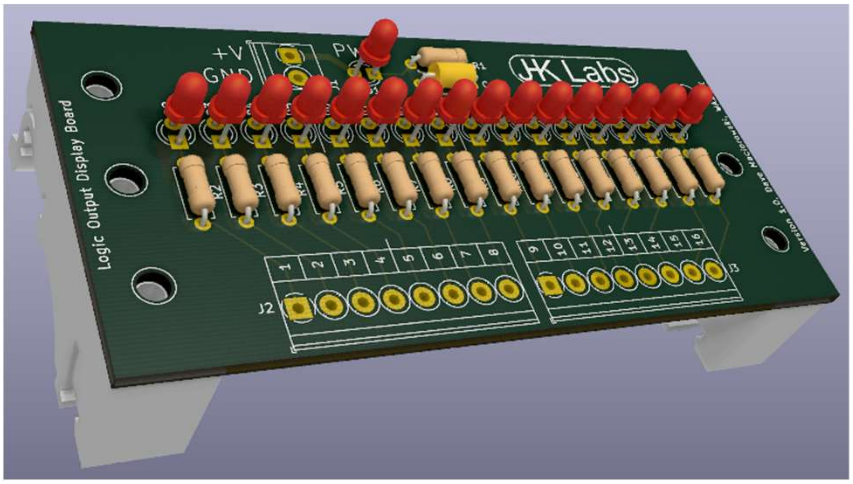
You can resize and spin to any view.

Use the Toolbar or click and drag.



I spun the view to look at the bottom. We didn't add Back Silkscreen, but we could have. The Green is the Soldermask. The Gray and Gold are the solder pads.

PCB Layout Editor



2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 111

Here’s a board I did that has the 3D models for the resistors, LEDs, capacitor, and the DIN Rail brackets on the bottom. I need to find or create models for the terminal strips.

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

That completes the schematic.
We're now ready to lay out the circuit board.

PCB Layout Editor

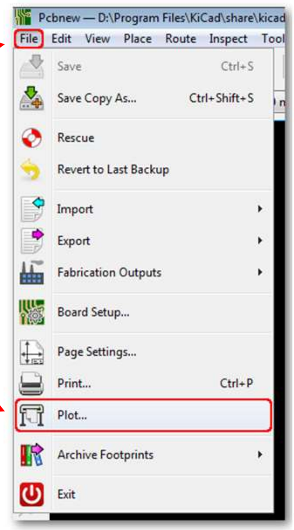
- Generate Gerbers and Drill Files
- Manufacturers Each Have Their Own Requirements
  - JLCPCB -- https://support.jlcpcb.com/article/149-how-to-generate-gerber-and-drill-files-in-kicad
  - OSHPARK -- https://docs.oshpark.com/submitting-orders/preorder-checklist/

To order boards, we need to generate Gerber and Drill Files.
JLCPCB conveniently has specific instructions on how to do that. I'll provide an overview here.
OSHPARK will do it for you. You just send the design to them.

PCB Layout Editor

Click File

then Plot...



We're going to Plot the Gerbers. Click File, then Plot.

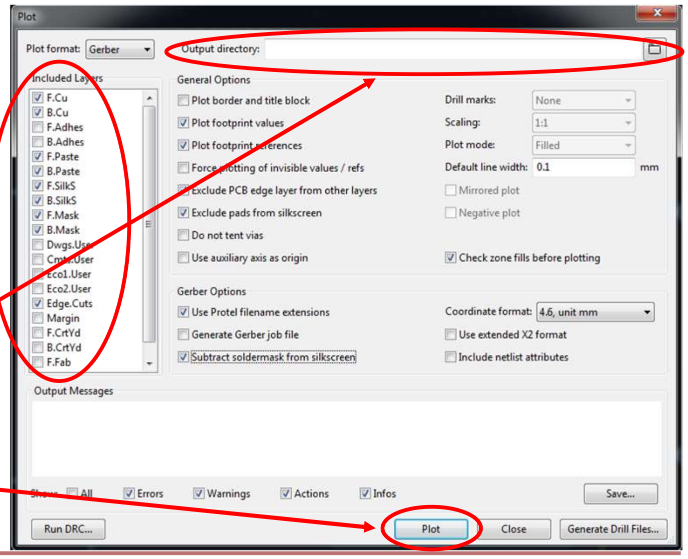
PCB Layout Editor

Included Layers Are Checked

Verify Checkboxes

Default Directory OK

Click Plot

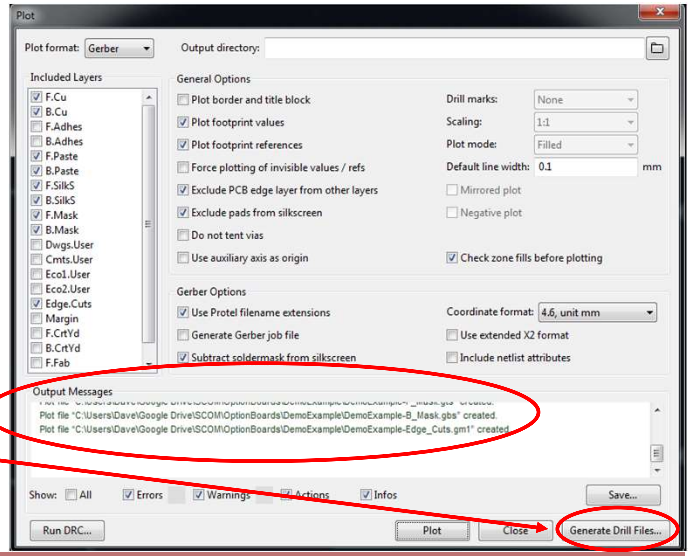


2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 115

The layers we need to include to manufacture a board are the Copper, Paste, Silkscreen, and Mask Front and Back, then the Edge.Cuts for the board size.
Carefully verify the checkboxes on this dialog. BE SURE TO CHECK AGAINST THE JLC WEB SITE, TOO. The default path Output Directory is fine, though I usually create a Gerbers directory. Click Plot.

PCB Layout Editor

Output Messages Verify Files Generated

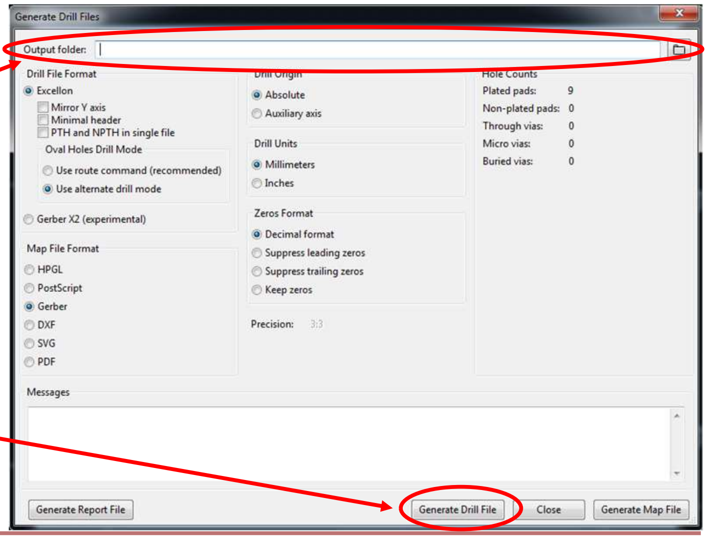


Click Generate Drill Files

Verify the Output Messages are all green.
Now generate the Drill files.

PCB Layout Editor

Default Same Directory as Gerbers

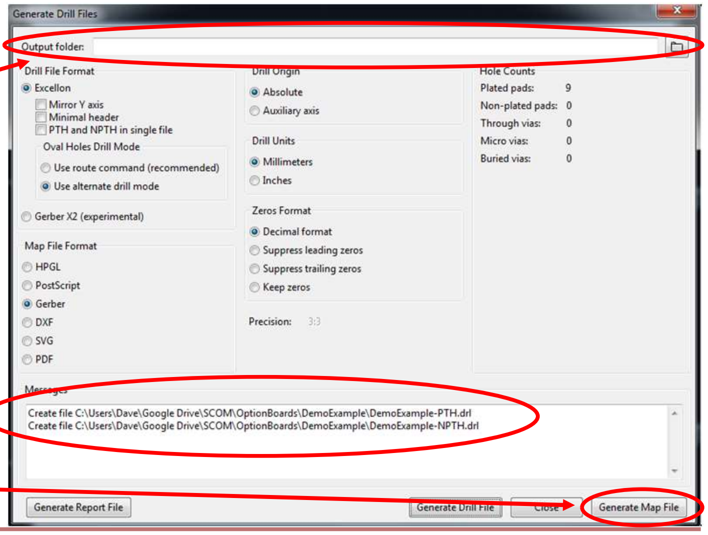


Click Generate Drill File

Again, default the output directory. Verify the selections in this dialog. Click Generate Drill File.

PCB Layout Editor

Default Same Directory as Gerbers



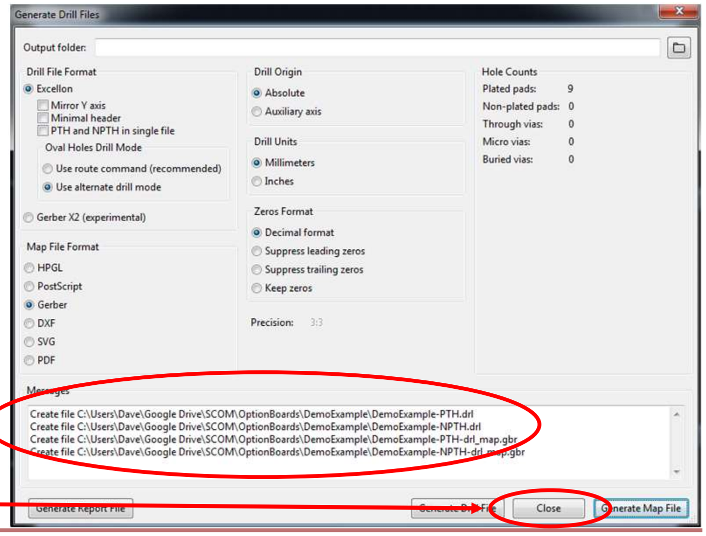
See Messages

Click Generate Map File

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 118

The Messages show that the Drill files have been generated. Click Generate Map File.

PCB Layout Editor



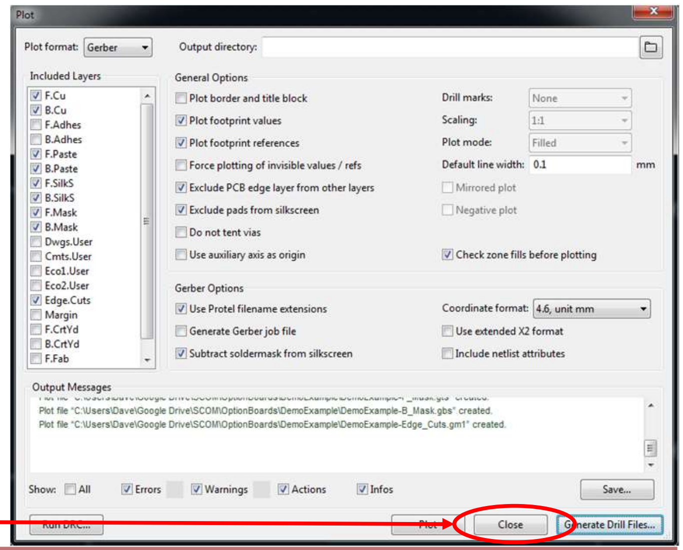
See Messages

Click Close

2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 119

Again, the Messages show the map files were generated. Click Close.

PCB Layout Editor



Click Close

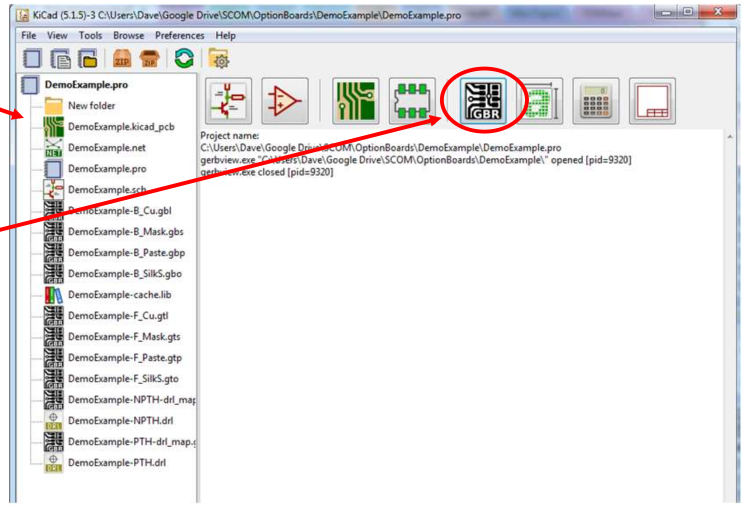
2/12/2022 Dave Maciorowski, WA1JHK -- JHK Labs 120

That brings us back to the Plot dialog. Click Close here too.

KiCad – Main

Output Files

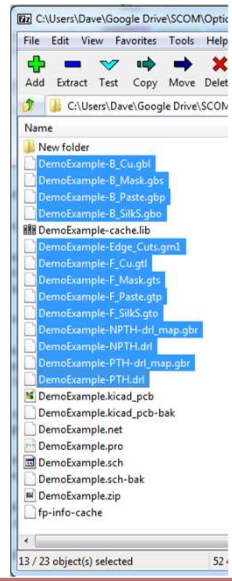
Gerber Viewer Looks at Output Files



Zip Gerbers and Drill

Select the Files and Zip them Up

DemoExample_Gerbers_220212.zip



Now we need to Zip the Gerber and Drill Files. I happen to use 7zip. Zip up the Gerbers and Drill files into your .zip file.
Choose a name for your file that makes sense.

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!!!