

Lab 4: Introduction to PCB Design

ECE 302, 302H

Motivation:

Breadboards are a great tool to prototype and test designs quickly. However, when a design needs to become more permanent and manufactured in a larger scale, engineers turn to PCBs (Printed Circuit Boards). In order to print circuits, manufacturers require files that specify your design. To accomplish this purpose, tools, such as KiCad, are used to layout a circuit on a board. Though the skills you will learn in this lab are specific to KiCad, many of the topics transfer to other industry software, such as Altium Designer.

PCBs consist of layers of copper alternated with layers of nonconductive material, typically a composite called FR4. The copper layers are etched (or milled) away such that only certain copper lines, called traces, are left. Certain locations have exposed copper pads so that components can be placed down on them. Drilled and copper plated holes, called vias, connect layers together.

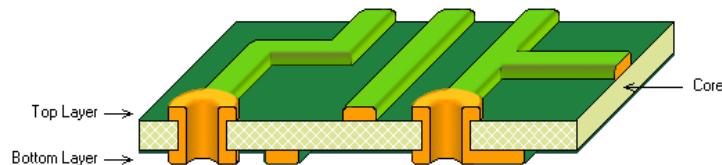


Figure 1 – PCB with FR4 “core”

In this lab, you are supposed to work in a group to finish the PCB.

KiCad Schematic Editor:

The schematic editor is the tool in KiCad that allows the engineer to define the circuit. This tool can be opened by clicking the “Schematic Editor” icon in the main KiCad window.



Figure 2 – KiCad Toolbar

For the purposes of these instructions, we will refer to the tool bar option from figure 1 based on the positional number from the top (1-23).

In order to place a component down on the schematic, click the tool-bar option 3 (or use the shortcut “A”). This should open up a menu to search symbols in your global KiCad library. The “symbols” are the figures placed in the schematic to represent the component. Sometimes, it is useful to use symbols outside the default list, this will require you to import a symbol into KiCad. For this course, we will not need to use this feature, but you can find quick documentation online.

Once you have selected a symbol, double click it and place it somewhere on your schematic (place by left clicking on the schematic).

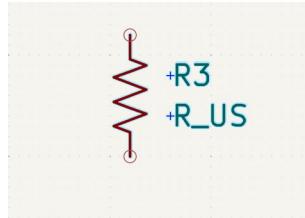


Figure 3 – Resistor placed down with default values

To modify the value of the component (eg. Resistance for a resistor), double click the symbol.

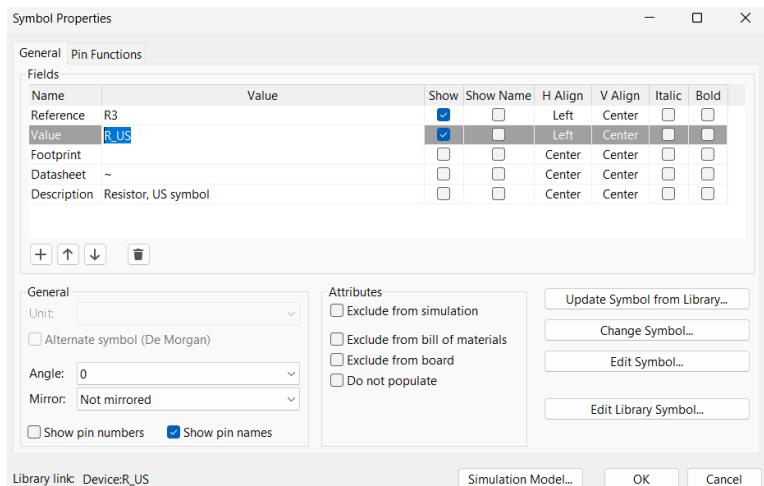


Figure 4 – PCB Symbol Properties Window

The symbol properties window allows you to attach a datasheet for the component and change the PCB footprint associated with the component. For now, adjust the value to match the desired value.

To connect multiple components together with a wire, simply hover over a pin. If this doesn't work, click on option 5 of the toolbar (shortcut "W") to activate the wire tool. Next, click two points to create a wire between them.

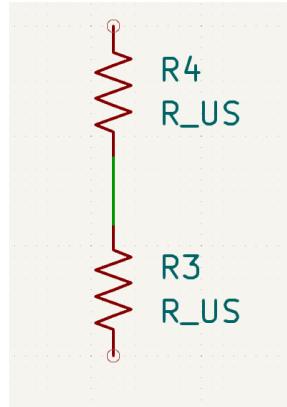


Figure 5 – Two default resistors connected together with a wire

It is often useful to label wires (nets) in your design. To add a label, click option **10** (shortcut “L”) to open the label tool.

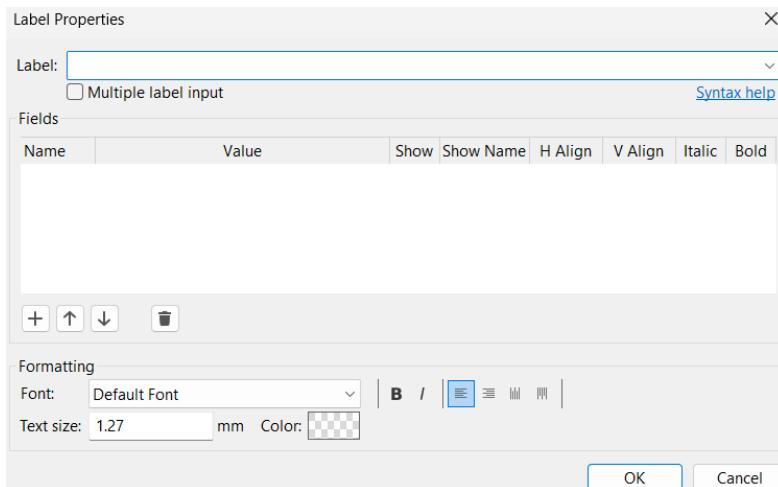


Figure 6 – Label Properties Window

This window offers several configurations for the label. If two labels have the same name, the schematic editor will assume the nets are shorted. Once the label looks correct, click “OK” and place the label on a wire in your design.

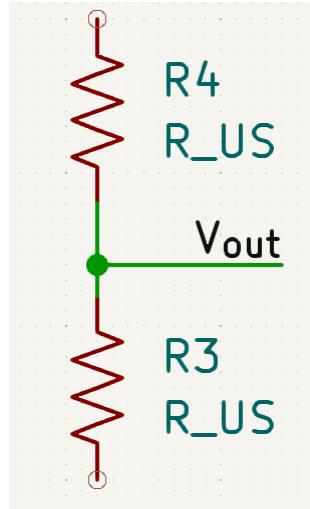


Figure 7 – Label placed at output of voltage divider

Conventionally, power nets (eg. Vdd and GND) are specified by specific symbols. Labels will serve the same purpose as these symbols, but if you want to place symbols, you can use option 4 (shortcut “P”).

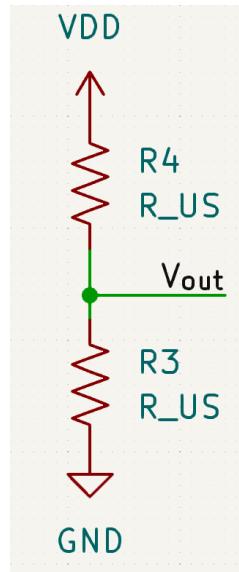


Figure 8 – Voltage Divider Schematic

Though the circuit has been specified, the schematic editor is also used for the next step of the design flow. Specifying the footprint of each component is important to link the symbol to how the component looks like on the PCB. The footprint is simply the space on the PCB reserved for a specific component.

To assign footprints, click on “Tools” at the top of the window and then click on “Assign Footprints”.

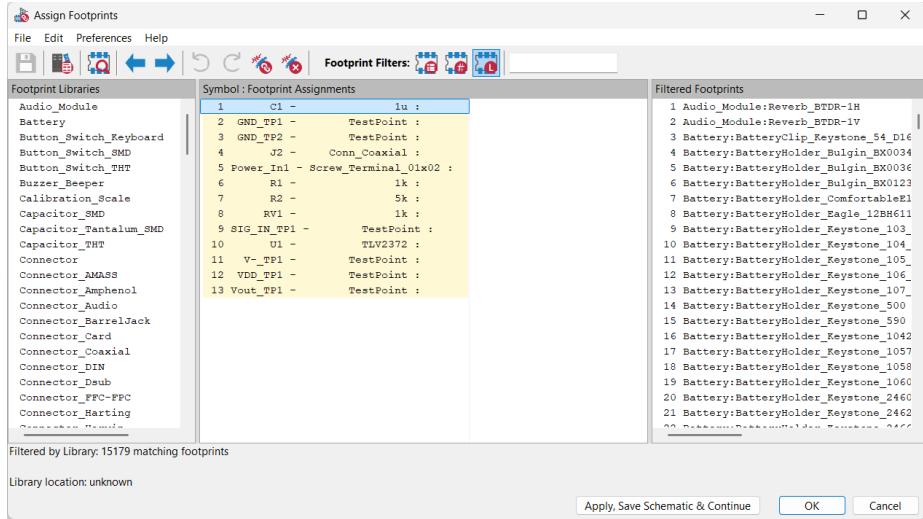


Figure 9 – Assign Footprints Window

The components highlighted in yellow are not yet assigned to a footprint for the PCB. Make sure all of them are assigned a footprint before moving on to the PCB layout.



Figure 10 – Footprint filters option at top of assign footprints window

Once you select the library on the left side of the assign footprints window, click the highlighted filter button from figure 10. This will filter out the footprints on the right side of the window based on the library. You can then input the specific part information on the search bar to the left of the filter buttons. The filtered footprints window should now narrow down to your search information.

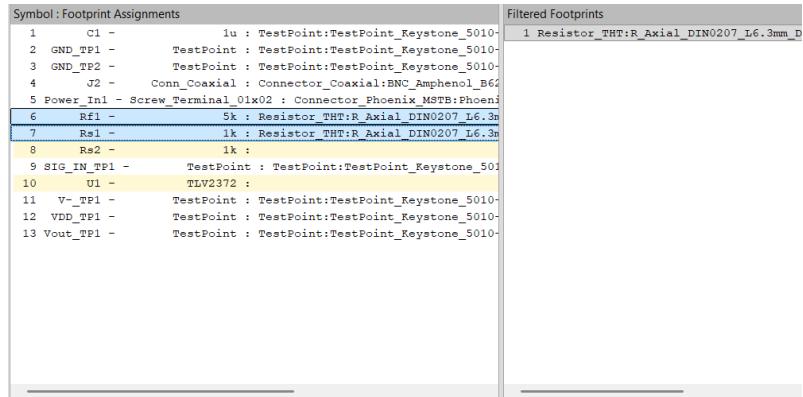


Figure 11 – Resistors highlighted to assign footprint

Once again, when you double click the footprint, the highlighted symbols will be assigned to that footprint.

If you have a footprint that is not in the default library, you can import a footprint file. We won't be using this feature in our labs, but it is an option that you can research more on your own.

Once you're done with the schematic, you are ready to update your PCB. Click on "Tools" at the top of the window and then "Update PCB from Schematic" (shortcut "F8").

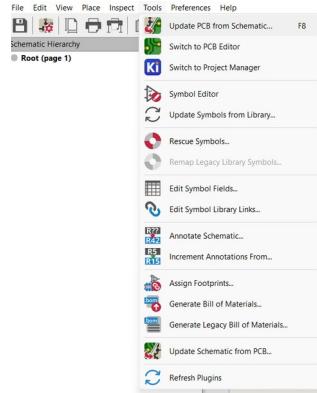


Figure 12 – Update PCB from Schematic option

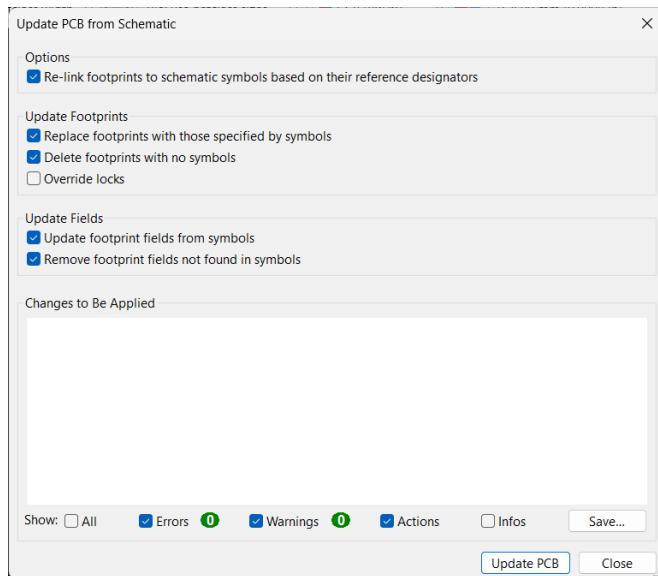


Figure 13 – Update PCB from Schematic Window

Select the options found in Figure 13 in the popup window, and then click Update PCB. Check for any errors in the textbox. If there are no errors, you are ready to move on to PCB editing.

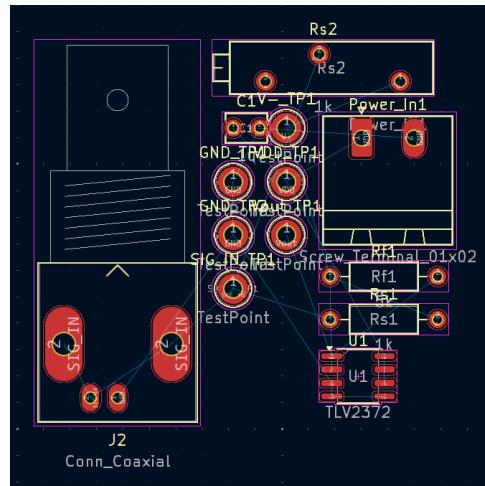


Figure 13 – Update PCB from Schematic Window

Figure 14 should show how the PCB window looks like when you have successfully imported changes to the PCB editor.

KiCad PCB Editor:

The first step in making a PCB is to position the footprints. There are several goals while doing this (eg. Minimizing area). However, you want to make sure you follow some guidelines:

- Place connectors and the potentiometer knob around the outer edge of the board, not the inner area.
- Place test points around the outer edge of the board, not the inner area. Space them far enough from other tall components (connectors) and each other so that you can easily probe multiple test points without the probes interfering with each other.
- You may end up with a dense circuit in the middle of your PCB with substantial “white space” around it. That’s okay – boards that are too small do not sit stably on the bench and make external connections more difficult.
- Do not try to lay out your circuit to look like the schematic. Instead, try to lay out components in a way that makes connecting them as easy as possible. As you advance in your understanding of EE, you will make layout decisions based more on electromagnetic and thermal considerations than on convenience.
- Place four vias near the four corners of the board, with hole diameter 3.4mm. These will be useful to attach “standoffs,” metal or plastic spaces that act like legs of a table.

Warning: This board includes a capacitor, known as a “bypass capacitor.” We’ll study the purpose of this later in the course. For now, you should simply internalize the rule that every IC should have a bypass capacitor across its power pins and as close as possible to the IC. This means that for our capacitor, it should be placed right next to the U1 chip.

Hint: You can place a component on the back side of the board by flipping it; click the component and use the shortcut “F” to do so.

Hint: To move a component, click and drag it. To rotate it use the shortcut “R”.

Use the text function (toolbar option #16) to add labels to your PCB. This is useful to specify where to place the components when you’re soldering them to the board. Make sure to set the “Layer” option for the text object on either the front or back “silkscreen”. This is the layer on the PCB that images and text are printed onto.

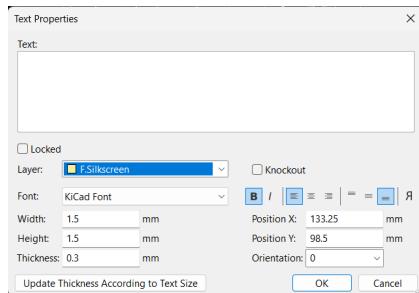


Figure 14 – Text Properties Window

It is best to label all your components, but make sure you at least label all your test points.

The last components to place will be mounting holes on the corner of your board. These holes can be used to mount the PCB or be combined with standoffs to keep the board upright when printed. The holes we will use should have a diameter of 3.4mm.

To add the holes (vias), use option #6 in the toolbar. Place the via down on a corner of the board and double click on it. This should open up the properties of that via. Make sure the properties match figure 16.

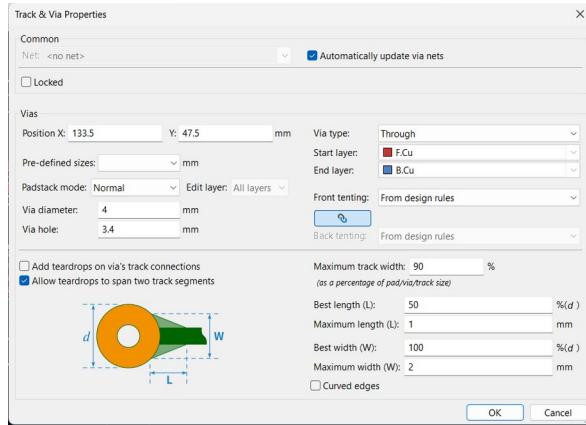


Figure 16 – Via properties menu

Place four of these holes: one at each corner.

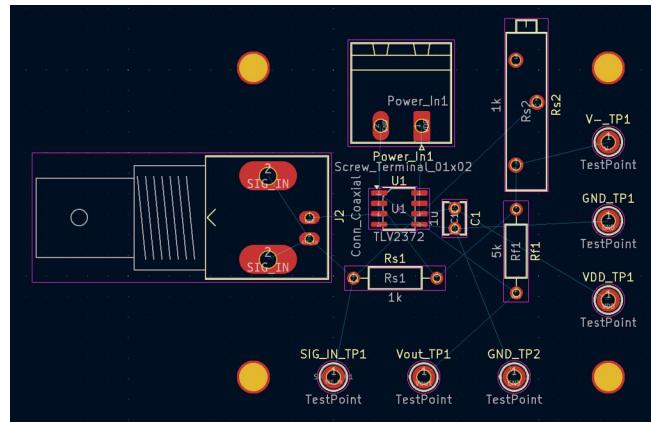


Figure 15 – Example placement of components with mounting holes

After placing down your footprints, you can draw an edge cut. This is the boundary of the PCB. To do this, select the “Edge.Cuts” layer on the right side of the PCB editor, as seen in figure 17.

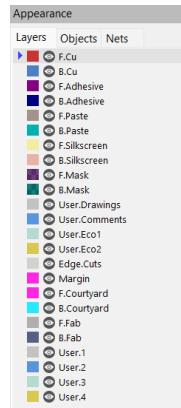


Figure 17 – Layer Selection Menu on right side of window

On the edge cuts layer, draw a rectangle with the toolbar option #11 tool. Make sure all the components' solder pads are within this boundary. It is also useful to place the connectors outside this boundary, so that the PCB isn't in the way of a cable connection.

It is often a good idea to fill a layer as the ground plane. This means that all grounded pads that pass through the layer will be shorted to one another. This also greatly reduces the number of connections you have to make on your PCB routing.

To make a ground plane, use the toolbar option #7 (make sure you have switched back to a copper layer in the selection menu. Once the tool is activated, click on a point on the edge of the ground plane. This will open up a window to configure this filled zone.

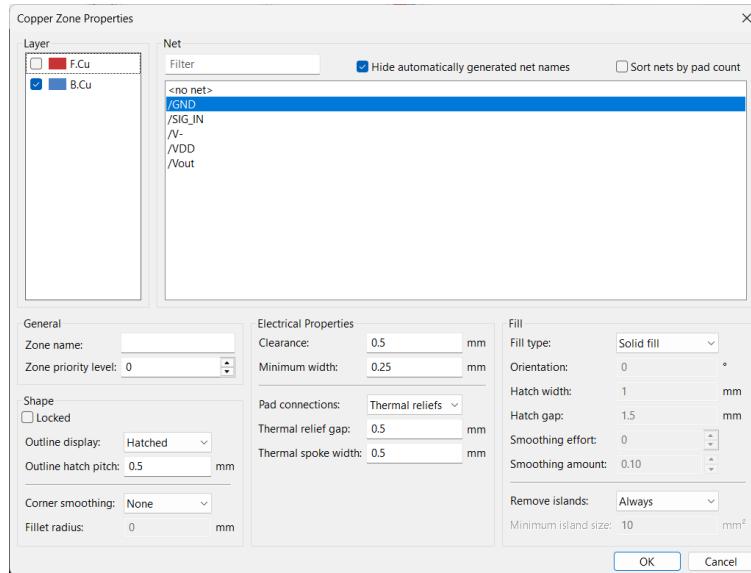


Figure 18 – Copper Zone Properties

For this project, make a ground plane on the back layer of the PCB using options in figure 18. After clicking "OK", finish drawing the ground plane. In this case, it is fine to have the ground plane line up with the previously drawn edge cut. Then, click "B" to fill the plane in.

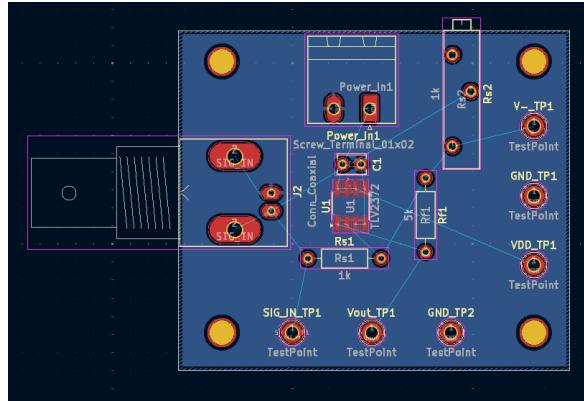


Figure 18 – Example placement of components with edge cuts and ground plane.

Now it is time for routing. Each thin line is part of the “rats nest”, which represents pads that need to be connected. The goal of routing is to eliminate all of the rats nest. To route your PCB, take one connection at a time. You can add a track by clicking option #4 on the toolbar or shortcut “X”. Here are some considerations for routing your components.

- Although the rats nest represents necessary connections, not every route you place needs to follow a line in the rats nest exactly. Use the rats nest as a starting point, but make the connections in the most efficient way you can.
 - The routes can be on the F.Cu or the B.Cu layer. To make a route run between layers, you can add a via in the middle of the route. This will create a hole to the back layer, which can be used for the rest of the connection. Add a via while routing a trace using the “V” shortcut key.
 - If possible, try to leave the ground plane on the back uninterrupted. This means run as few routes on the B.Cu layer as possible.

Every route should have an appropriate wire thickness. The exact thickness for any application can be determined based on several factors. There are many calculators online that take in context-based inputs, such as temperature and length of the trace, and output a necessary thickness (usually in units of oz/ft³). For this course, the lab documentation may provide trace thicknesses for you.

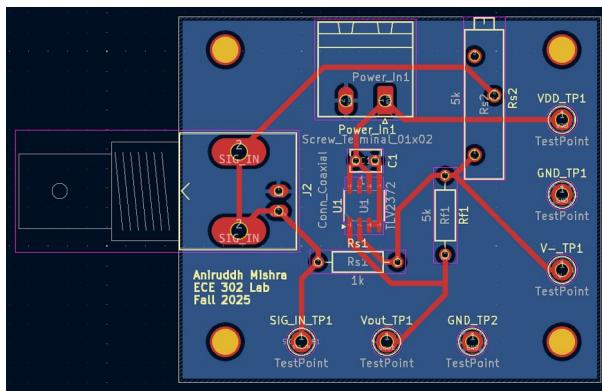


Figure 19 – Example routing of PCB

You can also add additional designs to your PCB. This can include pictures or silkscreen labels, such as your name. To add text, use the same toolbar option #16 as earlier.

The final step in PCB design is to make sure that none of the rules that are expected of the designer have been violated. This step uses a tool called the Design Rules Checker (DRC). Run the DRC by clicking “Inspect” at the top of the window and then clicking “Design Rules Checker”. This will open up the following window.

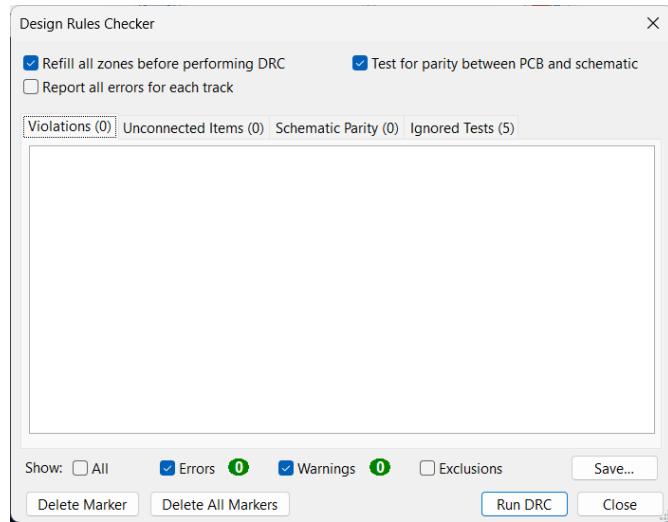


Figure 20 – Design Rules Checker

After clicking “Run DRC”, you should see the green “0s” next to the Errors and Warnings checkboxes at the bottom. If you see any errors or warnings in the textbox, go back and inspect what is causing the error. For example, figure 19 shows how the Rs2 component needed to be moved down compared to figure 18. This prevented a “Clipped Silkscreen” error.

To get a feel for what the 3d model will look like, you can click “View” at the top of the window and then click “3D Viewer”.

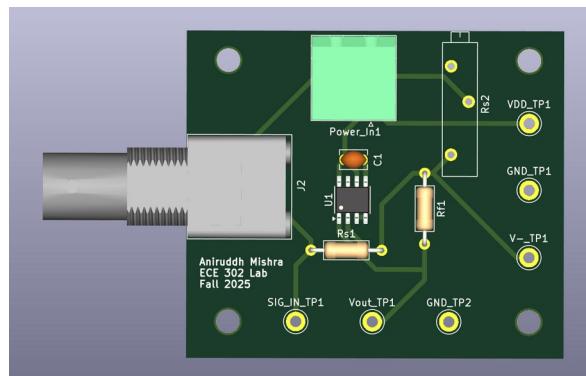


Figure 21 – 3D Model of PCB

Instructions:

Part 1: Designing the schematic of the circuit on KiCad.

This part of the PCB design process is to specify the circuit you want to lay out. Though the final PCB design doesn't require a schematic, placing components without previously planning connections is a great way to ensure a hardware debugging nightmare.

This schematic provided in this lab will be used for the variable amplifier lab we will do in the future. If you successfully design the PCB, you will be able to use it to complete the future lab.

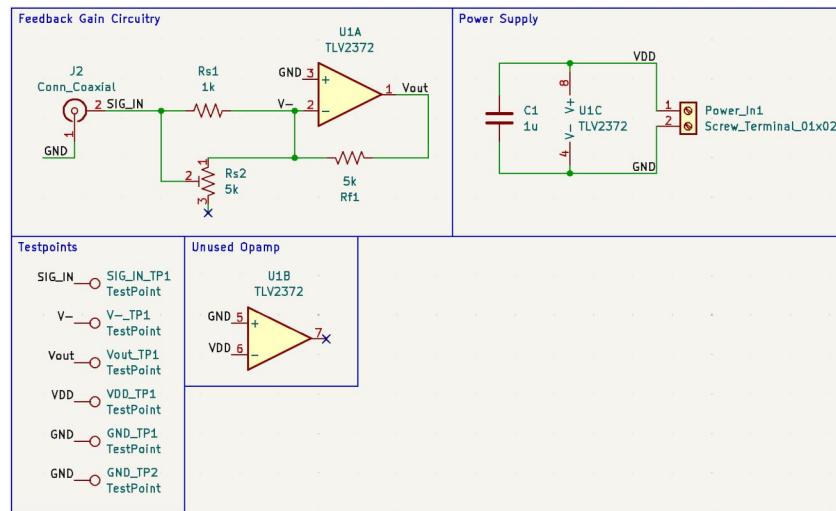


Figure 22 – PCB Schematic

The schematic you will be replicating can be found in Figure 22.

1. Create a new project with the default settings from the KiCad main window.
2. Open the schematic editor
3. Place the components found in figure 22
4. Wire the components to match the circuit above

Hint: If you want to add the text boxes found in the picture above, you can use toolbar option 19.

Note: The circuit schematic does not need to look identical to the example. However, make sure that all the connections are the same to ensure the same functionality.

Part 2: Assigning footprints

After you have designed the schematic, assign the footprints to each symbol. The footprints for each component can be found in the table below.

Component	Footprint
Rs1 and Rf1	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal
C1	Capacitor_THT:C_Disc_D3.8mm_W2.6mm_P2.50mm
Terminal Blocks	Connector_Phoenix_MSTB:PhoenixContact_MSTBA_2,5_2-G_1x02_P5.00mm_Horizontal
Test Points	TestPoint:TestPoint_Keystone_5010-5014_Multipurpose
Coaxial Connector	Connector_Coaxial:BNC_Amphenol_B6252HB-NPP3G-50_Horizontal
U1	Package_SO:SOIC-8_3.9x4.9mm_P1.27mm
Rs2	Potentiometer_THT:Potentiometer_Vishay_43_Horizontal

Once you have assigned all the footprints, accept the changes and “Update PCB from Schematic”.

Part 3: PCB Layout

Follow the documentation above to properly layout the components and route them on your PCB. Here is an example end result.

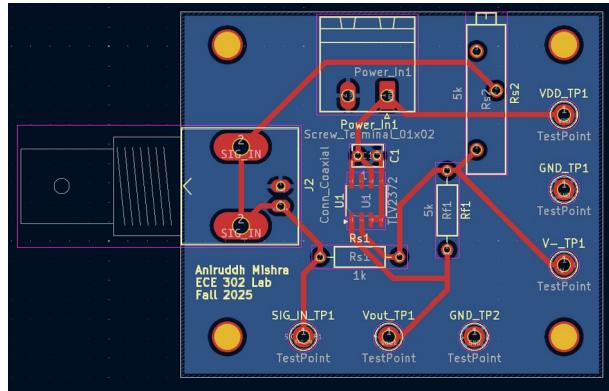


Figure 23 – Example routing of PCB

1. Place components and mounting holes in proper orientation
2. Draw edge cuts and ground plane
3. Draw traces until the rats nest is eliminated

Using the order of steps above, the PCB layout will be ready for the last step: sending out the order to the manufacturer.

Part 4: Placing images to make the PCB personal!

ECE 302 requires a lot of resources, and we are fortunate to have funding support from Texas Instruments and Würth Electronik. It is important to recognize sponsorship when you receive it, which we will do by adding their logos to our PCBs. Download the logos below or equivalent logos from the internet.



Figure 24 – Sponsorship Logos

1. Open the Image converter in KiCad from the main menu.



Figure 25 – KiCad Menu

2. Once the image converter window opens, click on “Load Bitmap” option, locate the logo and click “open”.

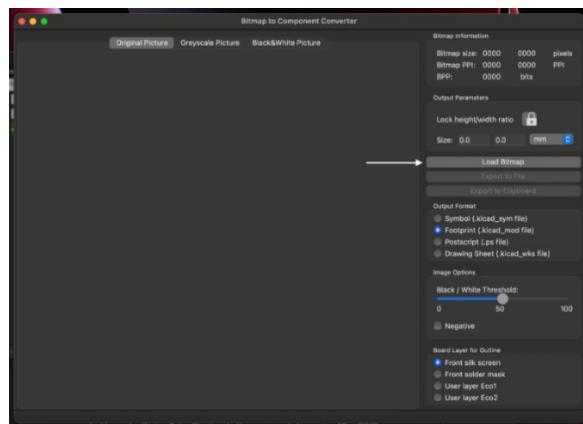


Figure 26 – Image Converter Window

- Once the image is loaded, adjust the size according to the space available on your board. And then select the options pointed out in the image below; you may have to change the sizing according to your design.

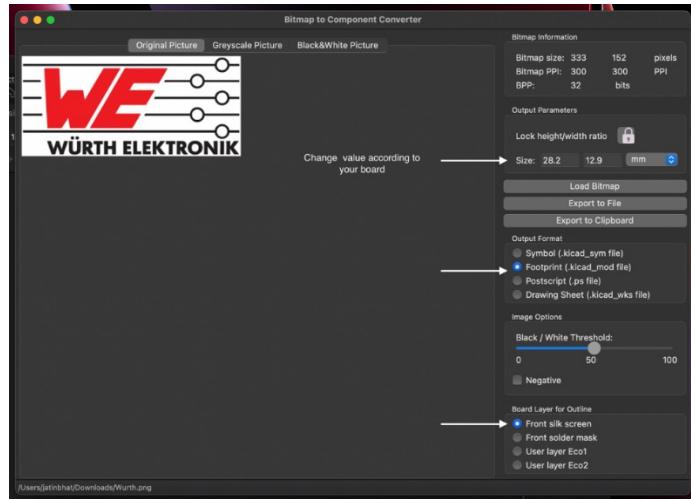


Figure 27 – Image configurations

- Now, create a folder anywhere, but perhaps inside the same directory as your project, and name it “images.pretty”. Now in the image editor, click on “export to file option”, locate the folder named “images.pretty” and save it inside the folder.
- Next open the PCB editor and click on Preferences -> Manage Footprint Libraries. Once the window opens, select “Global Libraries” option and click on the “+” button on the bottom left side. Next in the “Nickname” column, give a name of your choice (In this case it’s “graphics”). And in the “Library Path” column, locate the “image.pretty” folder you created previously and click on “OK”.

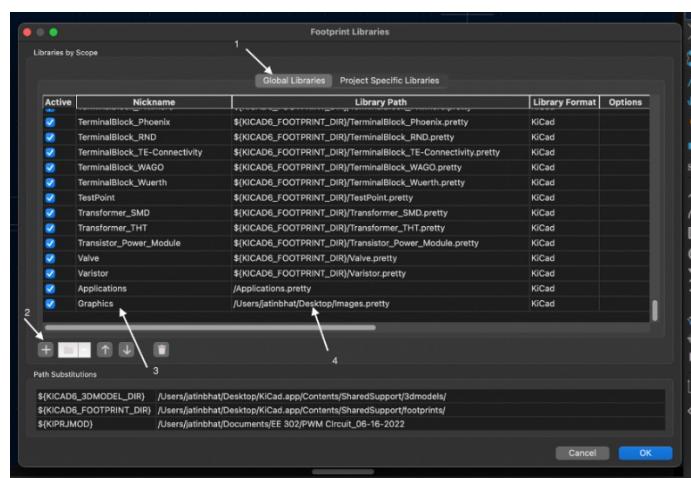


Figure 28 – Footprint Libraries Manager

- Now in the PCB editor, select option #3 in the toolbar.

- Once the window opens, search for the nickname you had given previously (in this case “Graphics”) and select the Wurth Logo. Click on “OK” next.

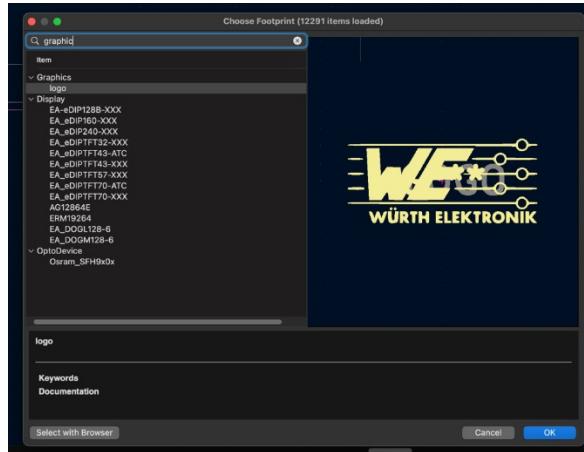


Figure 29 – Footprint search window

- The Wurth Logo will appear in the PCB editor and you can now place it anywhere. The final board should look something like this (with both Wurth and TI logos)

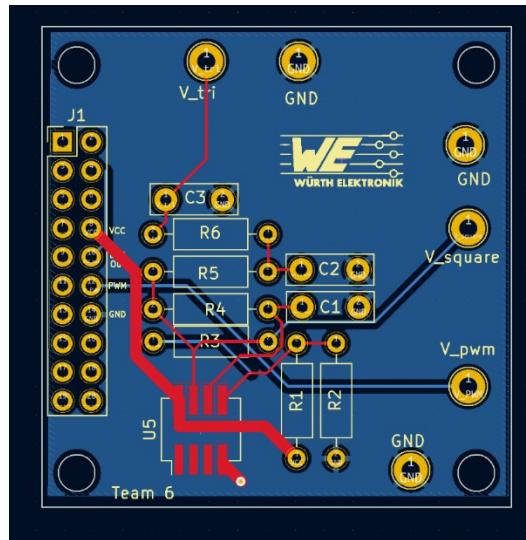


Figure 30 - Example of logo placement; not our circuit

You can add more images if you would like. Make sure that placement of the images doesn't sacrifice the quality of the overall design.

Part 5: Finalizing and ordering the design

1. Run the DRC tool. Instructions are at the end of the KiCad PCB Editor documentation.
DRC should end up clean.
2. Double check the following:
 - Mounting holes -- 3.4mm diameter (hole!) vias or cutouts at each corner with enough space from edge to accommodate spacer
 - Rectangular outline on the board cutout layer
 - Bypass capacitors near every IC across V+ to GND
 - Test points, ideally for every node, decently separated from each other, ideally along outer edge of board
 - Several test points for ground
 - Silkscreen labels for reference designators should be visible and easy to associate with a component, ideally even with the component installed
 - Silkscreen labels for test points with something human-understandable
 - Silkscreen name, month/year, Wurth logo, TI logo, other images you want
 - No DRC errors or warnings.
3. Make sure that your tent vias option is deselected. Go to “File” -> “Board Setup”.

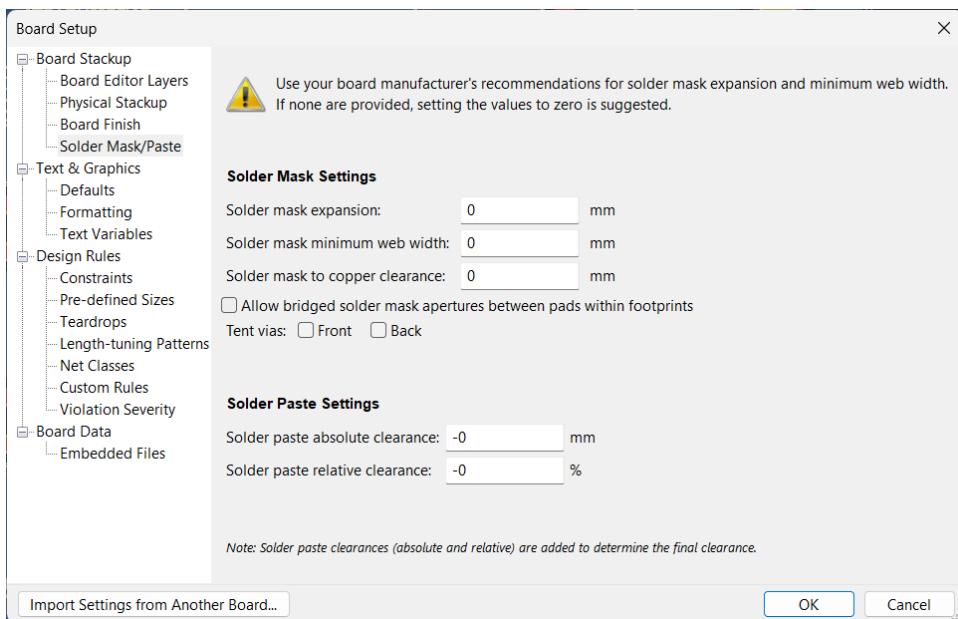


Figure 28 – Board Setup Options

Under the “Solder Mask/Paste” section, tent vias is deselected for Front and Back.

Warning: It is strongly recommended to have your TA inspect your layout before continuing.

4. Prepare to order your PCB
 - a. In the PCB editor, click File > Fabrication Outputs > Gerbers
 - b. Make sure that the following layers are included (you may need to un-check the Paste layers):

- i. F.Cu
 - ii. B.Cu
 - iii. F.Silkscreen
 - iv. B.Silkscreen
 - v. F.Mask
 - vi. B.Mask
 - vii. Edge.cuts
 - c. Check “Do not tent vias”
 - d. Check “Use Protel Filename Ext”
 - e. Choose an output directory (ideally a new folder – this process will create a lot of files)
 - f. Click “plot” – you should see a lot of “Plotted to...” messages in the output messages window
 - g. Click “Generate Drill Files”
 - h. In the new dialog box, ensure that the same output folder is selected
 - i. Leave all of the defaults as they are and click “Generate Drill File”
 - j. Locate all of the generated files (7 gerber files and one or two drill files). Select them all and compress them into a .zip file.
 - k. Name your .zip file with your last names and your lab section. For example, *Smith_Johnson_Fri9-11.zip*.
- 1. Go to JLCPCB.com and Create the account for your group.**
5. Click Add Gerber File and upload your .zip file. This should take you to an ordering page
 6. Choose the following
 - a. Base material = FR-4 (default)
 - b. Layers = 2 (default)
 - c. Dimensions = (enter the actual dimensions of your board if they’re not detected automatically)
 - d. PCB QTY = 5 (default)
 - e. Product type = Industrial
 - f. Different Design = 1
 - g. Delivery Format = Single PCB
 - h. PCB Thickness = 1.6mm (default)
 - i. PCB Color = Choose whatever color you like!
 - j. Leave the remainder as their defaults
 7. When you are ready, click “save to cart.” Look at the cart and make sure that your order is there and that your names are present.
- 8. Place the order and pay for the PCBs.**

After placing the order, keep checking your email and correcting any issues raised by JLCPCB. Ask your TA for help if necessary.

Do not delete your .zip file, and be sure to share the project with all project partners.

What you need to submit:

- 1. Screenshot of your final layout.**
- 2. Screenshot of your JLCPCB order page to show the order has been successfully paid.**

Please order PCBs as soon as possible. PCBs will take 2 weeks to arrive.

You will not get your PCB on time for the soldering lab if you place the order too late!