# KiCAD For Beginners

By Zuhayr :)

Hey guys! So, I've put this little document together for those of you who need a quick and clear way to understand how KiCAD works, since Justin hasn't been the greatest with lectures and teaching material. These notes are for KiCAD 6.0, which I highly recommend you use, if possible. Hopefully this should somewhat get you to pass Assignment 4.

Good luck!

(Disclaimer: I do not know everything about KiCAD, I'm a student, just like all of you. I apologize if there is something that is missing from these notes, I am just sharing what I know, hope it helps!)
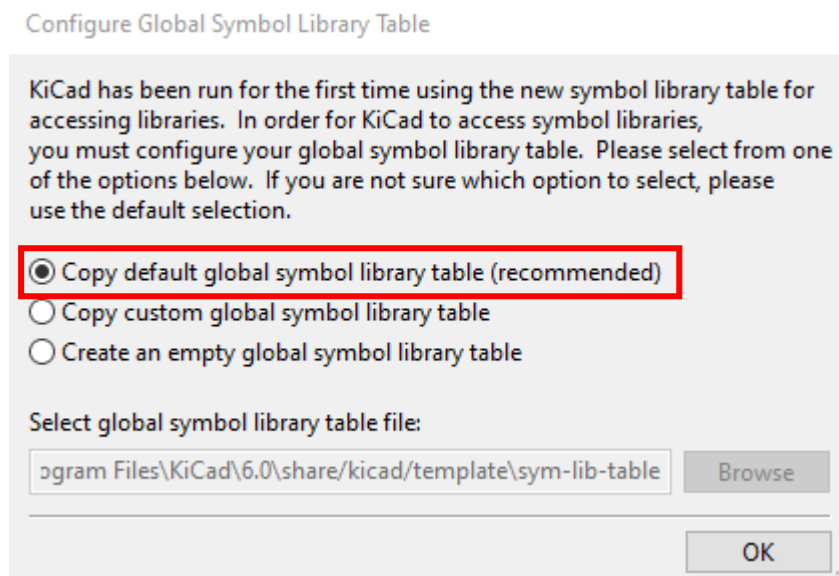
Part 1: The Schematic

1. When opening KiCAD, you should be greeted with this screen. For MEC1003F, we'll only be making use of the Schematic Editor, PCB Editor and Gerber Viewer.
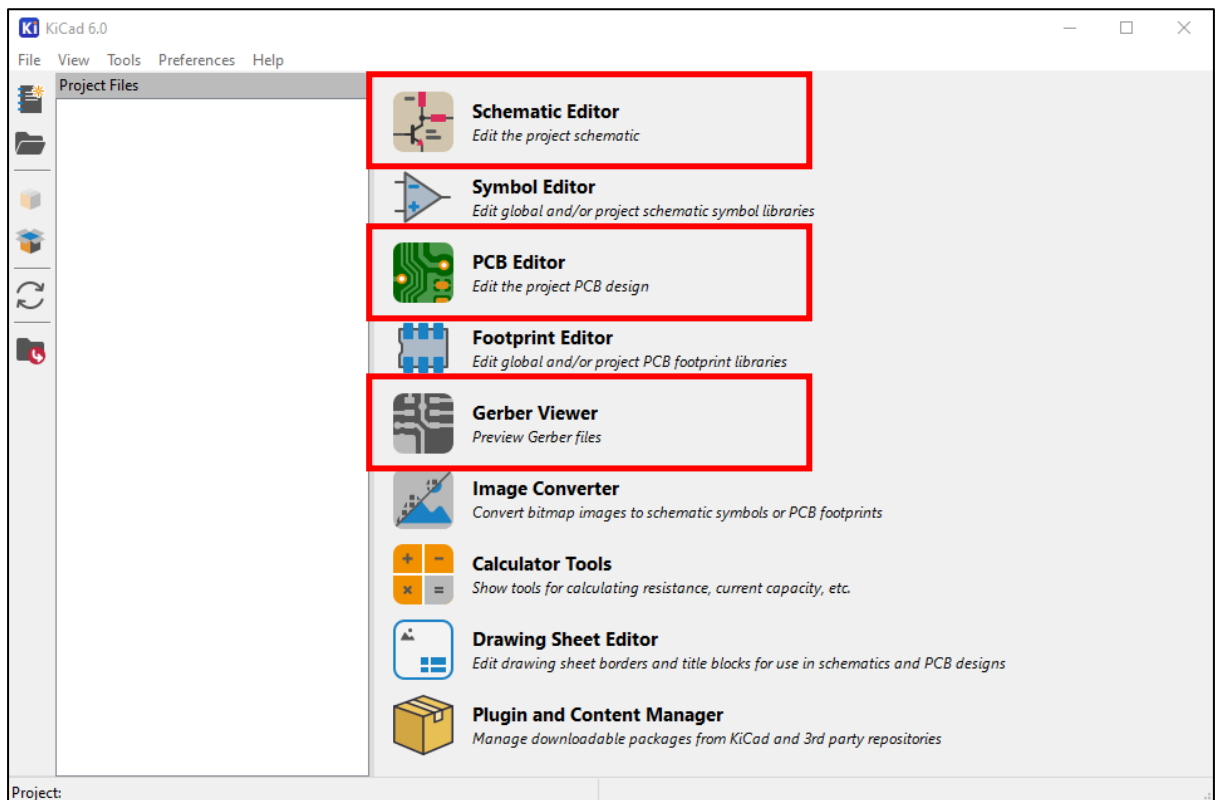

2. To begin a new project, click on the 'Create new blank project' button, on the top left corner of the screen.
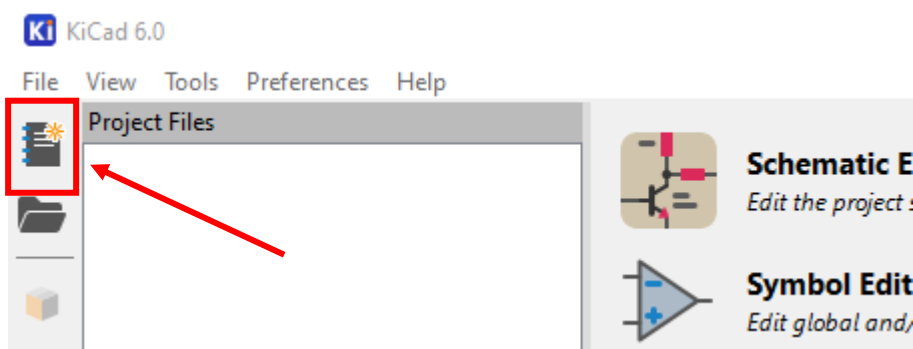
3. Save your KiCAD project file wherever you need to, with a suitable name. Marks are assigned to the naming of certain files later on, so make it a habit to name files clearly.

4. Once the new project is created and saved, you can now click on Schematic Editor to begin designing your actual circuit. If a popup, such as below appears, select "Copy default global symbol library table" and click OK.

5. You should now be brought to the Schematic Editor. To start things off, select "Page Settings" on the top toolbar. This is where we'll fill in the title block of our schematic. This is very important, as marks are assigned to this specifically.
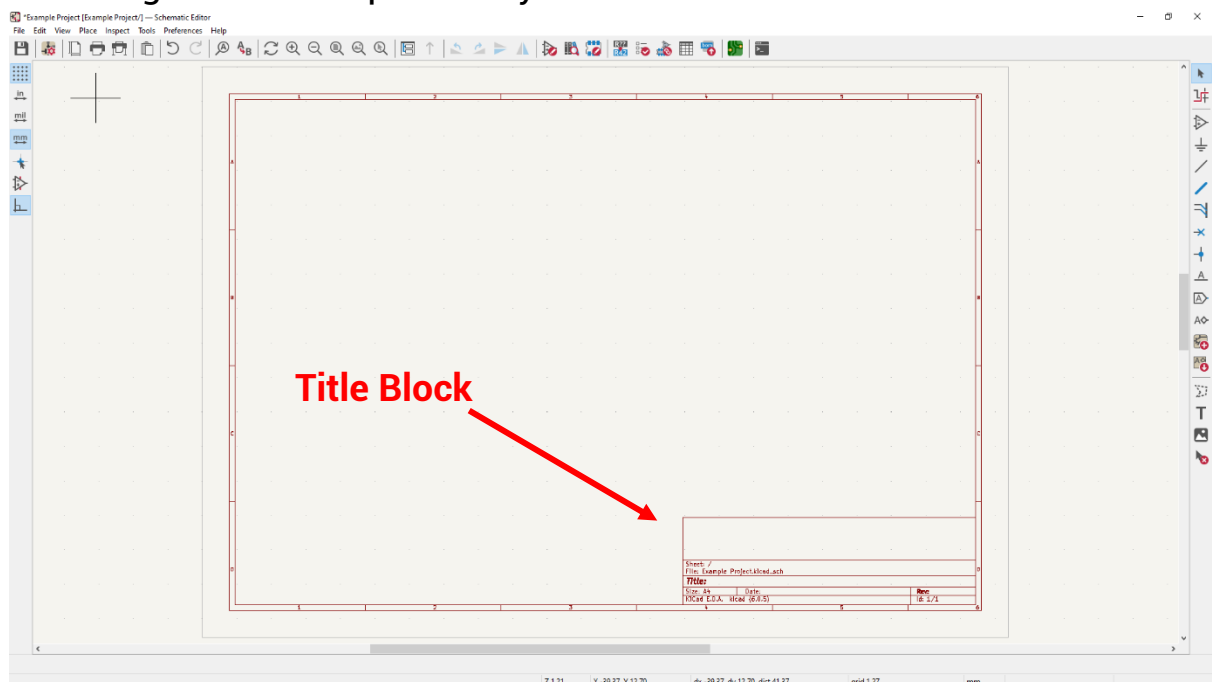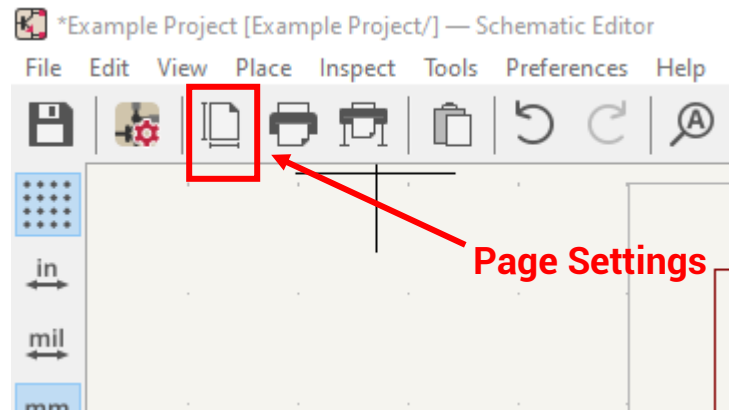
6. In Page Settings, we need to make sure that an Author Name, Circuit Name, Date and Revision Number are all included. I have included examples of what these could possibly be. (In order to automatically add the current date into your title block, click the "<<<" button.) Put the Author Name and your student number as shown below, so as to make sure they appear at the top of the title block. Markers are very specific about this.

Example of a Title Block:

- The circuit should be named after what it is used for
- Revisions have a lowercase "v", followed by a number. Just use v0 for the assignment.
- Author should be your name

7. Now for the actual schematic, we are usually given a rough sketch of a circuit to build. In order to add symbols into our schematic as they are seen on the sketch, press the 'A' key to (A)dd a symbol. All the symbols in the given sketch should be included in your schematic. You can search for various symbols in the popup window that appears.
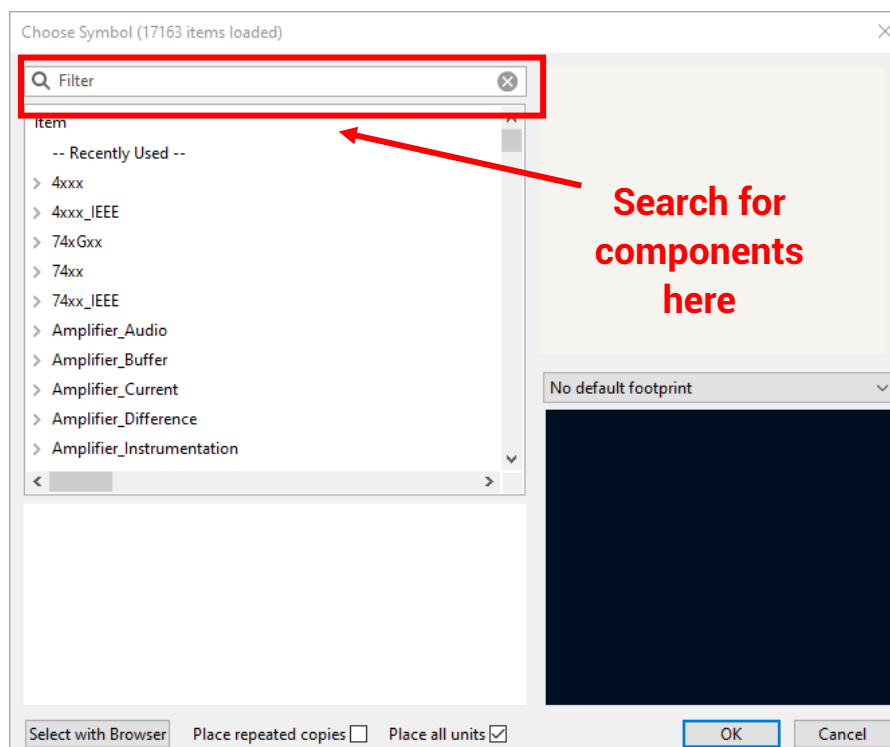
Here is a list of common symbols that you may need to use and their names in KiCAD:

- Resistors: R_Small_US
- Ground Symbol: GND or GNDPWR
- Input Power Symbol: This is normally whatever is given in the sketch, such as VCC, VSS, VD, etc.
- Capacitors: C_Small
- Connector Pin Headers: Conn_01x(number of pins)_Male. For example, a 3-Pin Connector would be Conn_01x03_Male, etc.
- Any other specific symbols will be given to you, such as transistors, etc.

8. After selecting the symbol, simply just click wherever you want to place it on the schematic. While placing a symbol, you can click the 'R' key to rotate it as necessary. To move a symbol after it has been placed, select it and press the 'M' key.

9. Once you have placed all your required symbols, they need to be connected using wires. You should wire the circuit exactly as shown in the rough sketch given to you. You can add wires by pressing the 'W' key to enter (W)iring mode.
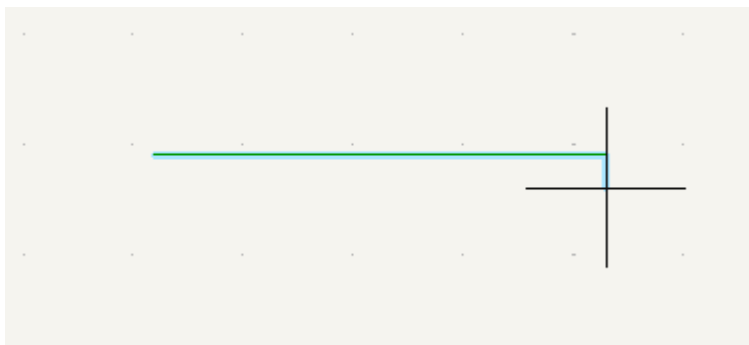

10. Click once to start placing a wire while in wiring mode. Double click to end the wire, making sure all your components are properly connected. To exit wiring mode, press the Escape key.
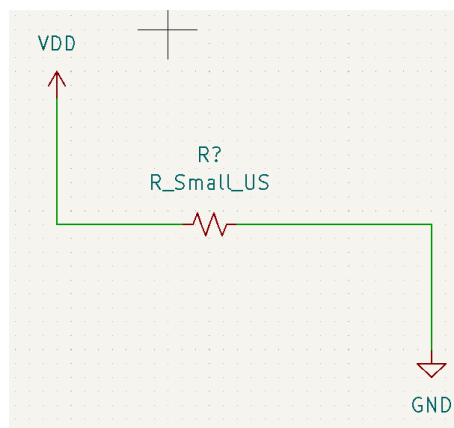
11. Annotating your schematic is also very important. This means that all your components are numbered (resistors are R1, R2, R3 and capacitors are C1, C2, etc.) and that they are given values (1k resistors, 10nF capacitors etc.). To automatically number all components, click on the "Fill in schematic symbol reference designators" on the top toolbar, and click "Annotate."
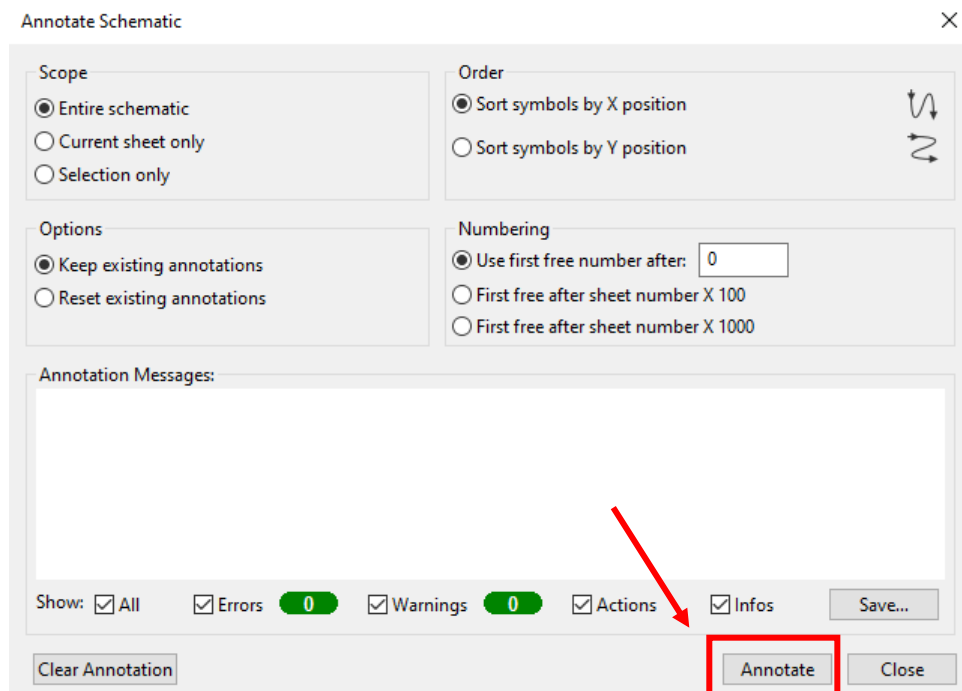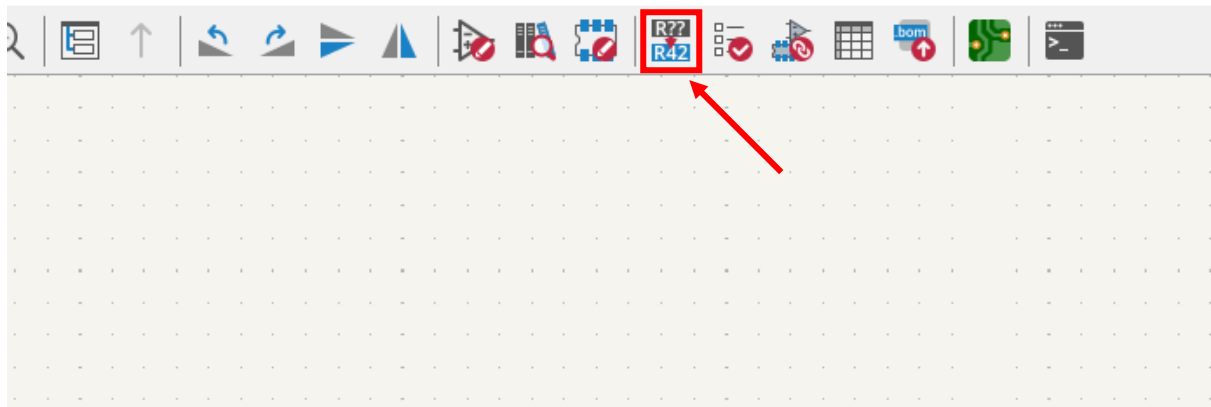
12. All your components should now be numbered. In order to allocate values to certain components, double click the name of the component in the schematic and enter the value. (Some components, such as transistors and pin headers do not need values and should not be renamed). Additional labels can also be added using the 'L' key, if necessary.
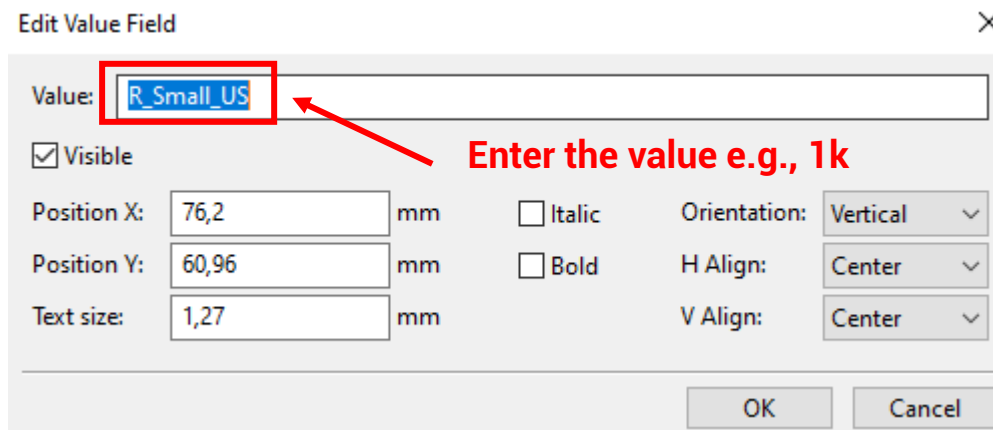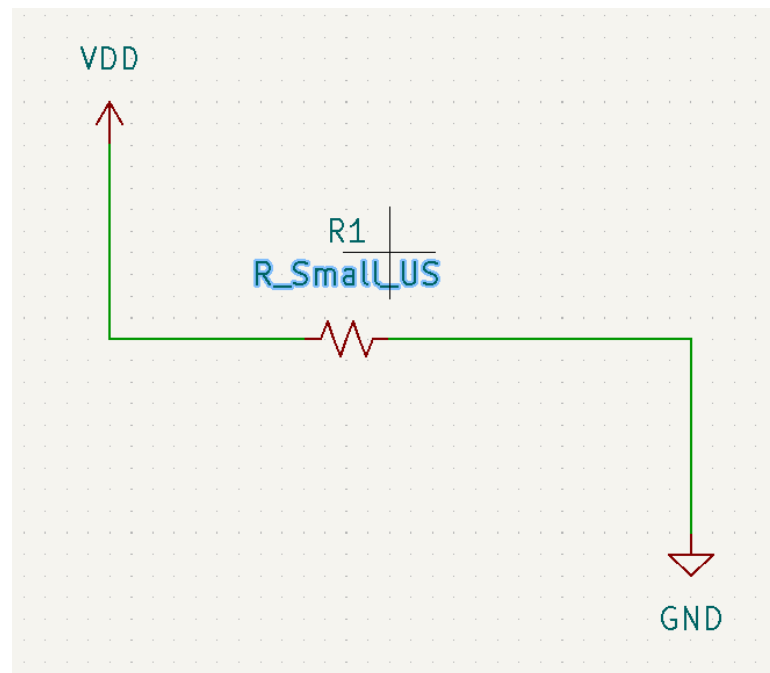

13. Congratulations! Once all your components are properly annotated and have the correct values, you should have a fully complete schematic! Now we have to take this schematic and use it to design our PCB…

Part 2: The PCB

1. Ironically, the first step of designing your PCB actually starts in the Schematic Editor. In order to import all your components to the PCB Editor, all components require a Footprint. Essentially, a Footprint of a component is what the component will physically look like on the board and in real life. Most components do not have a footprint automatically assigned to them. In order to do this, start by double clicking the symbol you wish to assign a Footprint to.
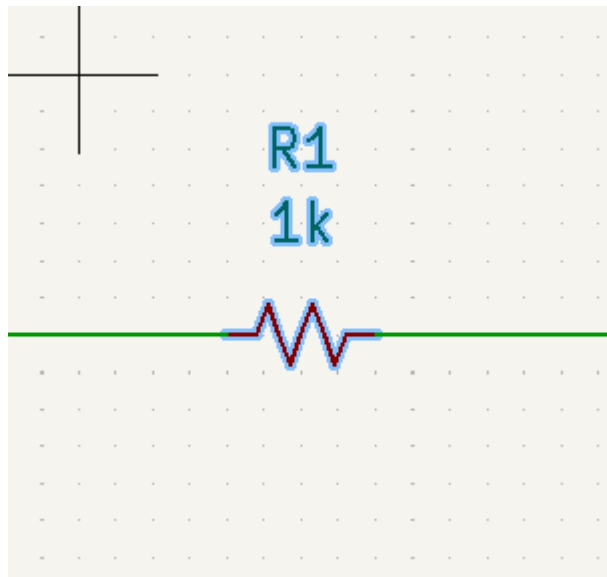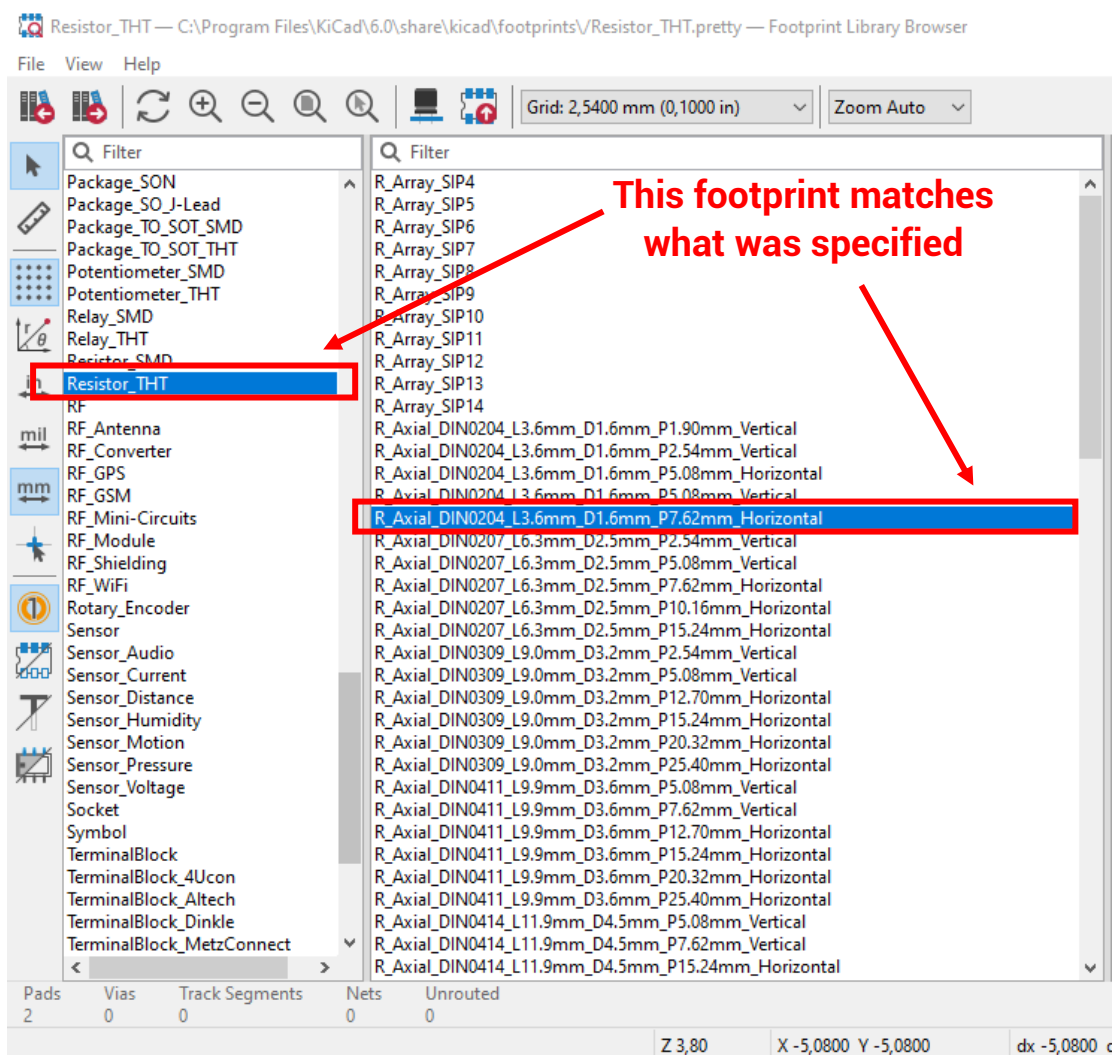

2. In the Symbol Properties page, click on the three lines in the Footprint box.
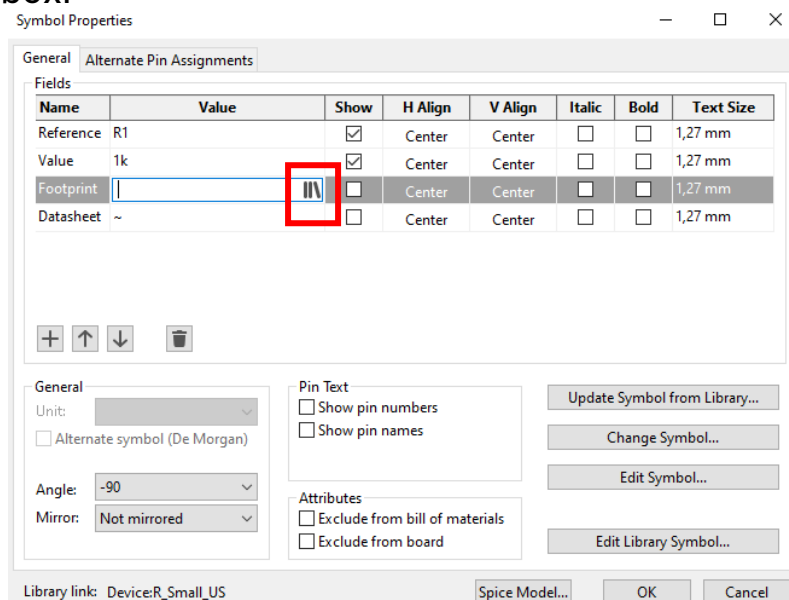
3. The Footprint Library should now be open. Usually, the assignment will specify what footprints to use for your components. Let's use a resistor as an example.

This is the footprint we need to use for our resistor, in this example

This footprint matches what was specified

4. Double click on the footprint once you have found it in the library and click OK.
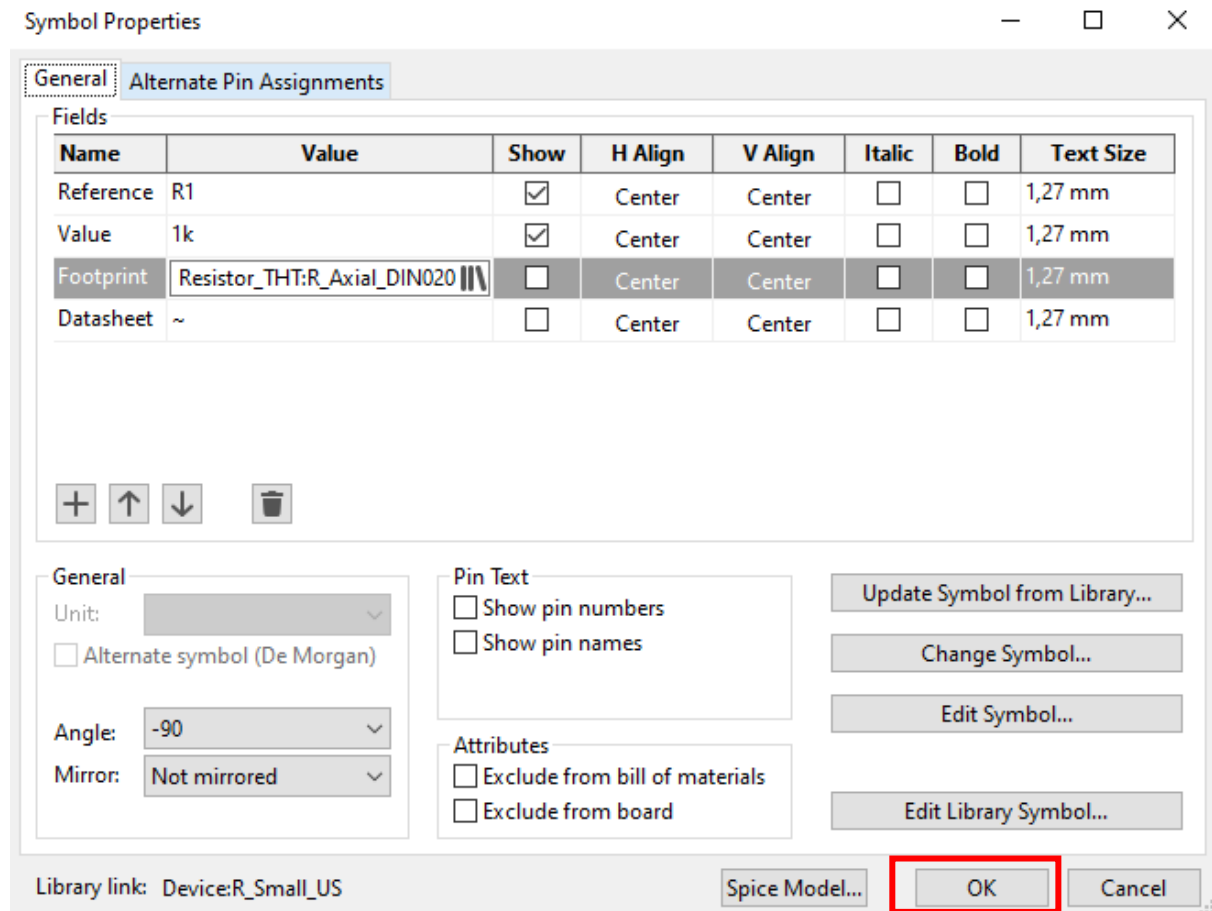
5. Repeat the previous step with all components that do not yet have the correct footprints. This is obviously crucial in ensuring that your board has the correct components on it.
6. Once all footprints have been assigned, we can now close the Schematic Editor and open the PCB Editor. This can be done by opening the PCB Editor from the first screen, or directly from the Schematic Editor using the "Open PCB in board editor" button on the top toolbar.
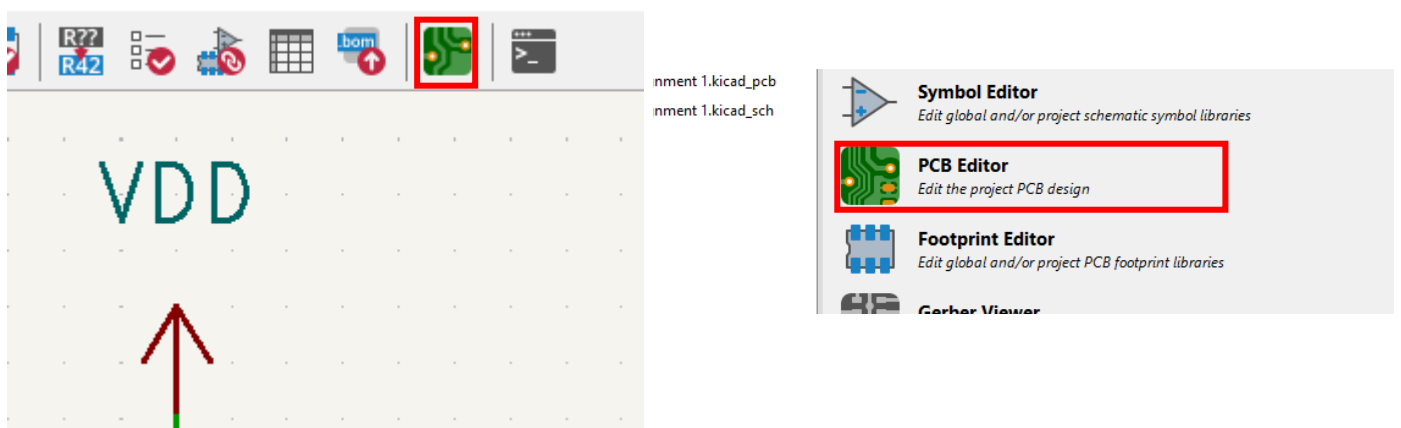
7. To import all our components into the PCB Editor, click on the "Update PCB with changes made to schematic" button in the top toolbar, or by pressing F8. Click on "Update PCB" and click to place all your components down. (You can also update your components after placing them, if necessary. This could be if you change a footprint or a value in the schematic after you have made your PCB.)


Import schematic
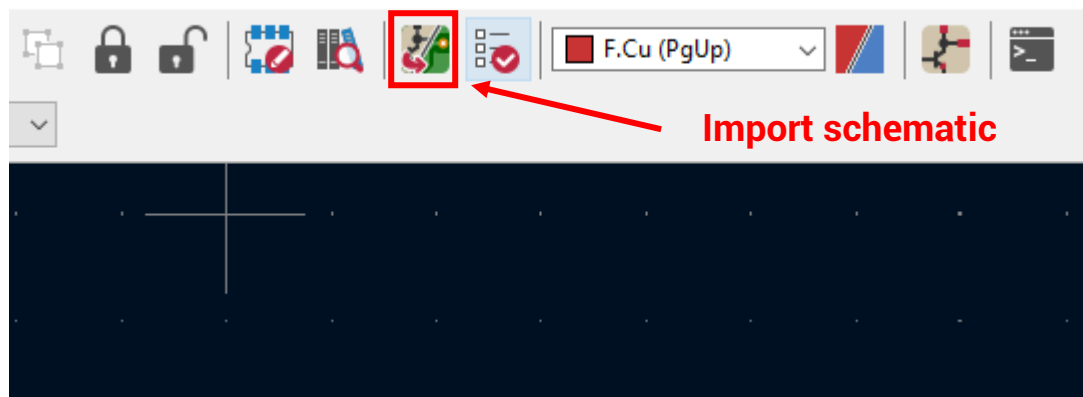
Update PCB from Schematic

Options
☐ Re-link footprints to schematic symbols based on their reference designators
☐ Delete footprints with no symbols
☑ Replace footprints with those specified in the schematic

Changes To Be Applied

Processing symbol 'J1:Connector_PinHeader_2.54mm:PinHeader_1x03_P2.54mm_Vertical'.
Processing symbol 'J2:Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical'.
Processing symbol 'Q1:Package_TO_SOT_THT:TO-92_Inline'.
Processing symbol 'R1:Resistor_THT:R_Axial_DIN0204_L3.6mm_D1.6mm_P7.62mm_Horizontal'.

Total warnings: 0, errors: 0.

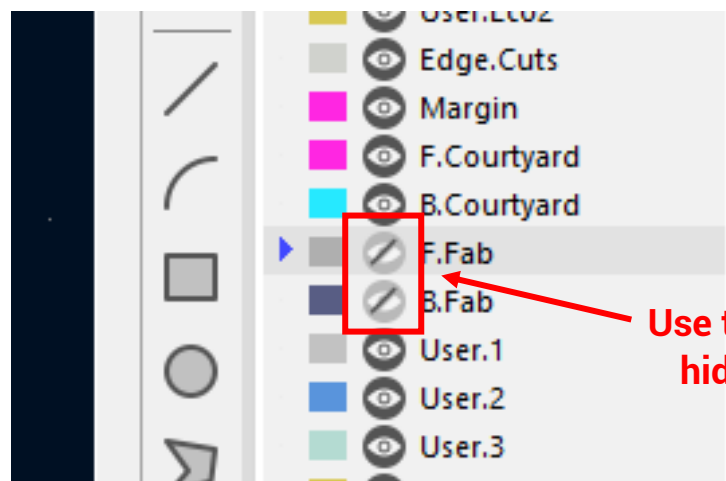Show: ☑ All   ☑ Errors 0   ☑ Warnings 0   ☑ Actions   ☑ Infos   Save...

Update PCB   Close

You should now have a very messy PCB design, something similar to the picture below.
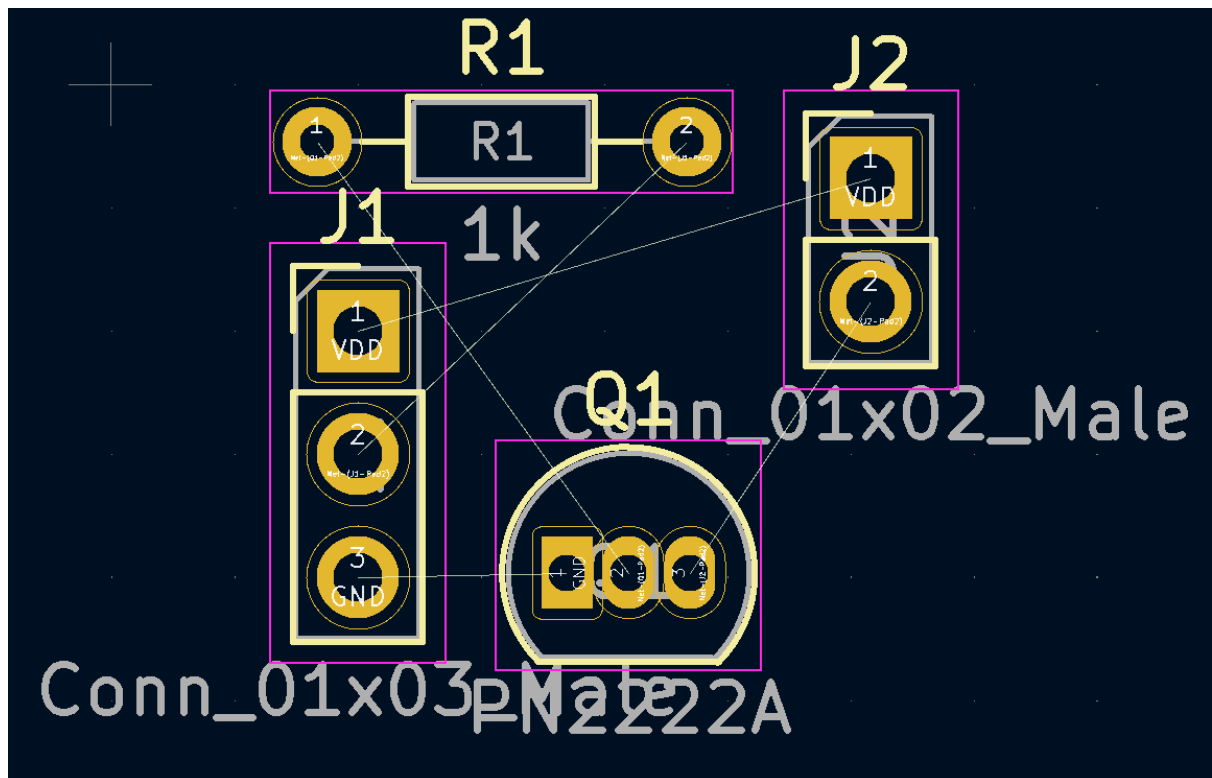
To reduce clutter while working on the PCB, hide the F.Fab and B.Fab layers on the layer panel on the right hand side of the screen.

Use these to show and hide various layers

8. Now we have to properly and neatly arrange our components on the board. Similar to moving and rotating components in the Schematic Editor, we can do this in the PCB Editor as well.

There are some basic conventions to follow when deciding on the layout of your board:
   - Place the pin headers near the edges of the board and space them apart from each other. This makes wiring easier when the board is used after it is printed.
   - Do not have large, excessive empty spaces on the board, but still space your components out. More empty space means more material must be used, leading to a higher PCB manufacturing price.
   - Components should not overlap each other.
   - Place components so that the wiring is as concise and simple as possible. The thin white lines between components indicate connections via wires.

Here is what a good board layout would look like:
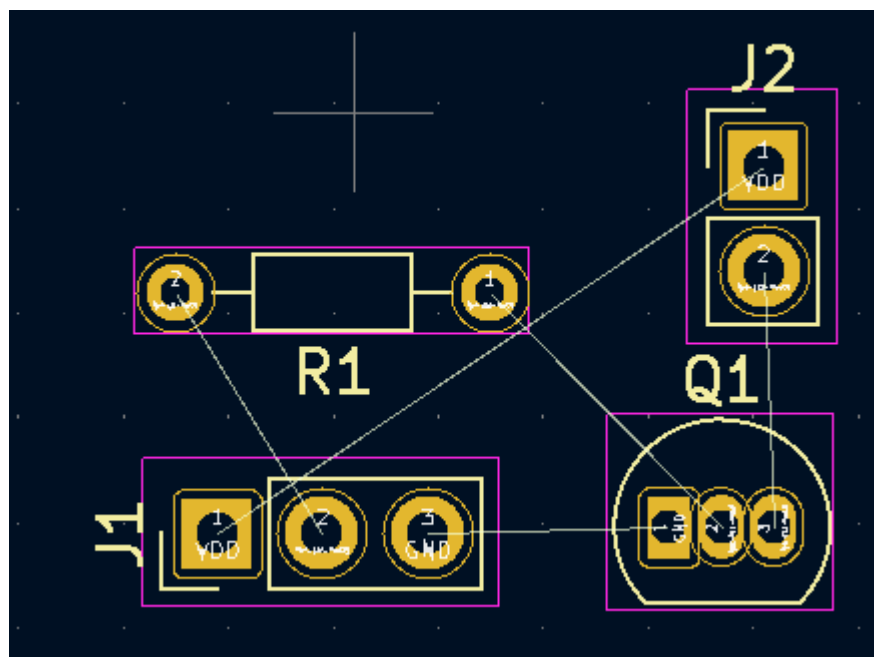
9. The board does not yet have a physical border, we need to create that manually. Do this by selecting the "Edge.Cuts" layer in the layer panel, and drawing a rectangle using the "Draw a rectangle" button on the right-hand toolbar. Click around your components to make create the outline of the board. Remember to switch back to the "F.Cu" layer when this is finished.

Indicates that Edge.Cuts is selected


Draw a rectangle

Outline
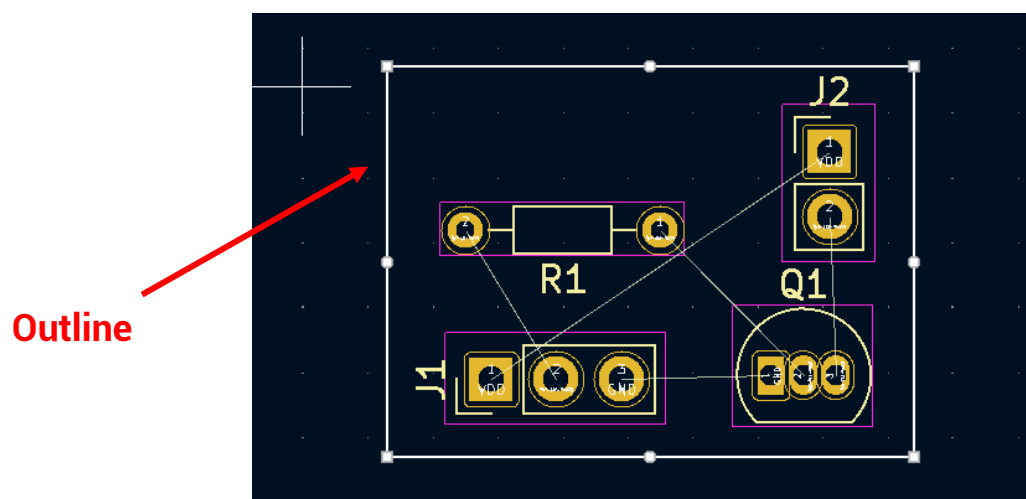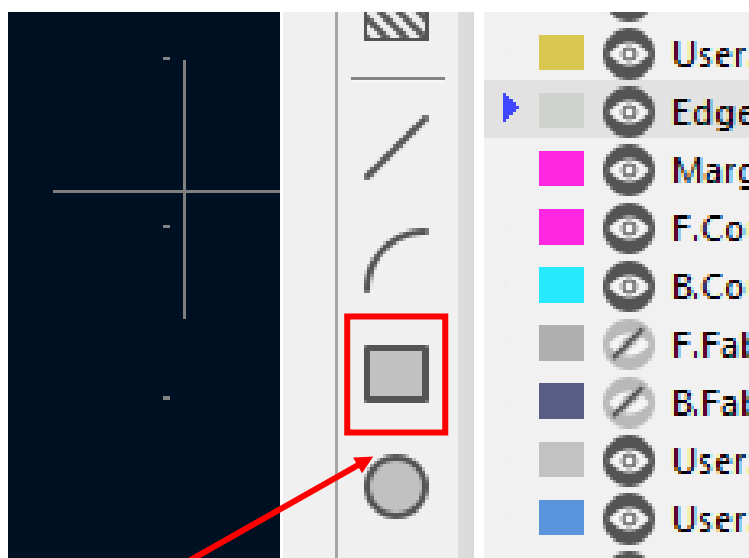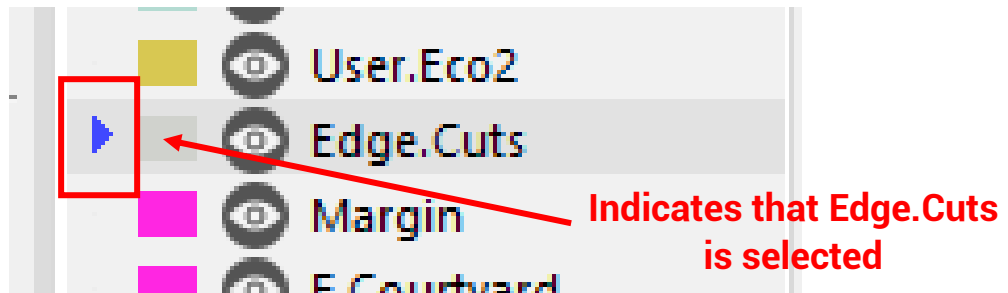
10. Now we need to 'route tracks' or connect our components by using tracks within the board. But first we need to define the width of these tracks, as it is critical that we use different track widths for different connections in the circuit. Places in the circuit where we expect a large amount of current should have a thicker track.

    To do this, click on the "Track" dropdown menu on the top toolbar, and select "Edit Pre-defined Sizes…"
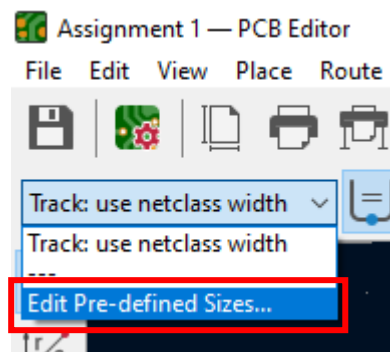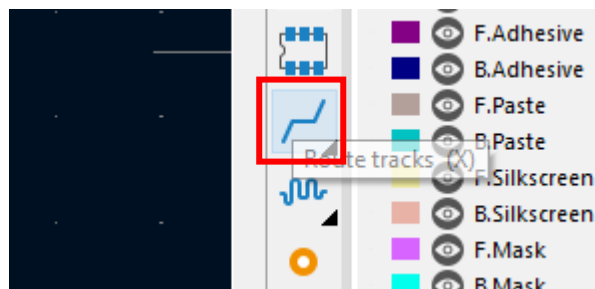
11. A page with 3 columns should appear. Click the + button under the "Width" column and add a track of width 0.4mm and another of 0.6mm. These are the sizes recommended by our tutors.
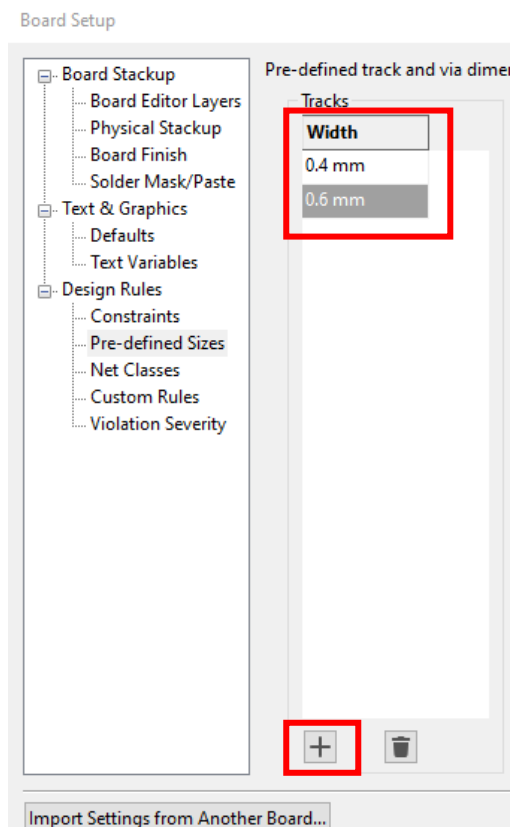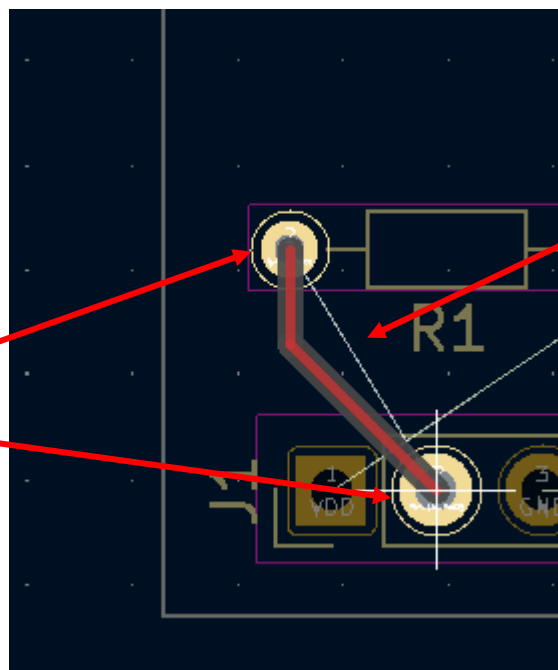
These sizes can be selected by using the same dropdown menu on the top toolbar.

12. We can now route our tracks. Select the "Route tracks" on the right-hand toolbar. While in routing mode, select one of the 'holes' on the board. These act as nodes for routing connections to be made. Connect these nodes such that most of the white lines indicating connections disappear. (Remember to select the correct track width!)
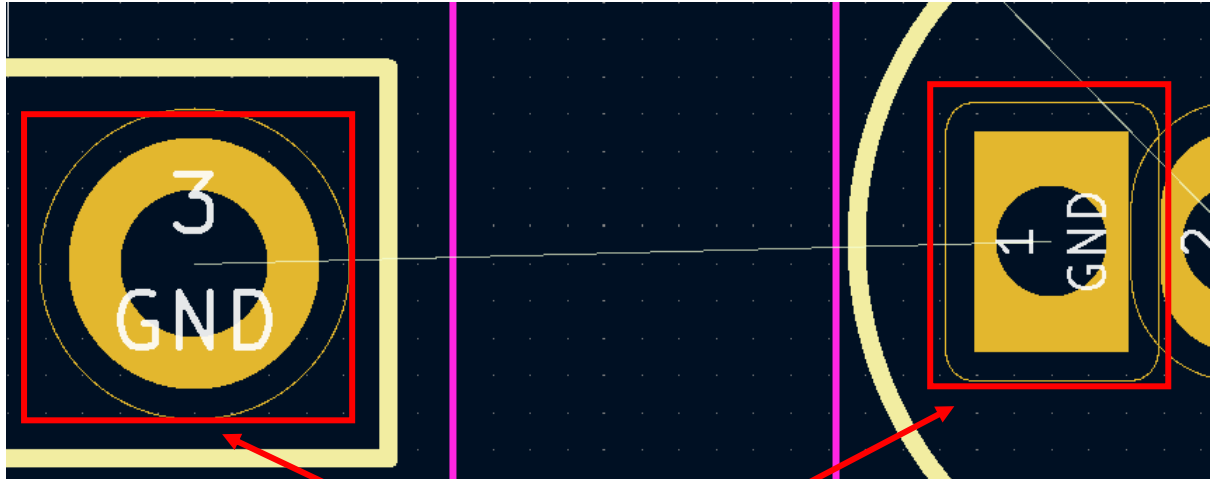

White line indicating that a connection must be made

Routing the track to connect these nodes
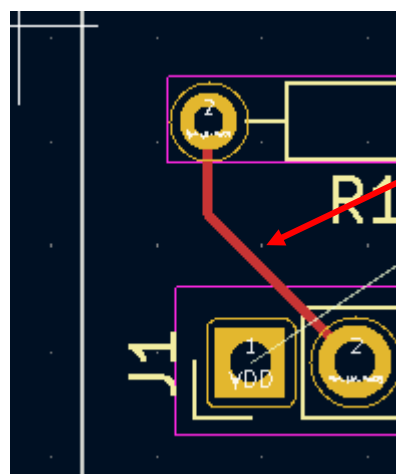
White line disappears when track is placed

13. Do NOT route any nodes that are assigned to GND or GNDPWR. This will be addressed later on. If you zoom in on a node, it should have white text which shows where the node connects to.


Do NOT route these

14. If you need some of your routes to cross over each other, you can make use of a via. A via is essentially just a hole that connects a route from the front of the board to the back. While placing a route, press the 'V' key and then place your route through a via. Press 'V' again and place a route to return to the other side of the board if necessary. This can make tracks a lot simpler.
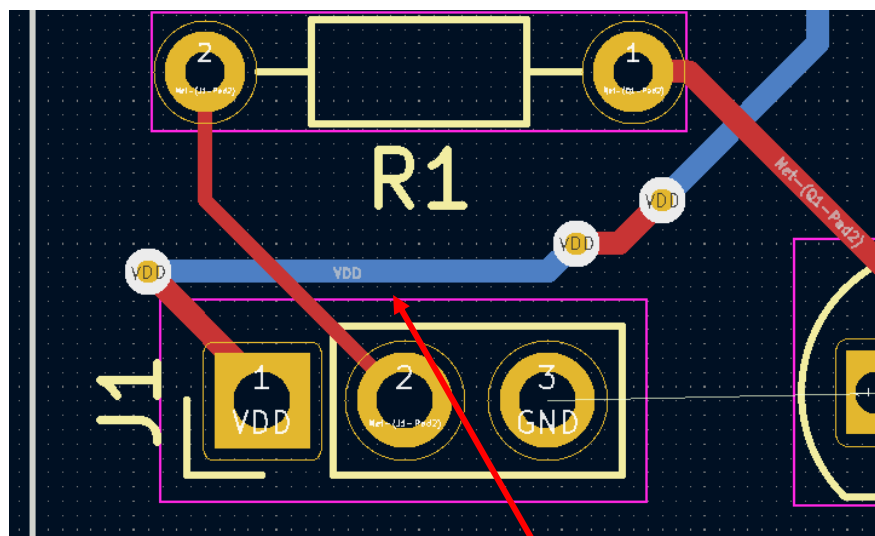

Blue routes indicate routes on the back, while red indicates routes on the front

15. Once all tracks have been routed (except for those going to GND), we can now start out polygon pour. A polygon pour is basically a layer of copper on the front and back of the board that we can use to assign to ground or GND. This is why we didn't route those nodes earlier, as the polygon pour will automatically connect to those nodes.

   To begin the pour, select the "Add a filled zone" button on the right-hand toolbar.
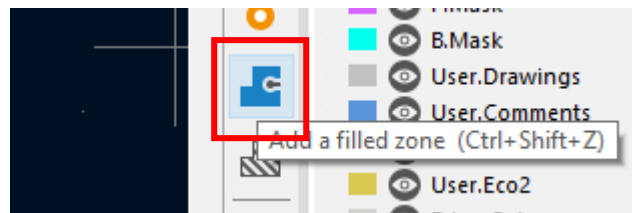
16. Click on one of the corners of the outline of your board. A properties page for the copper layer should pop up. Select GND or GNDPWR as the net for the layer and click OK. (Make sure only F.Cu OR B.Cu is selected, both layers need to be separate)
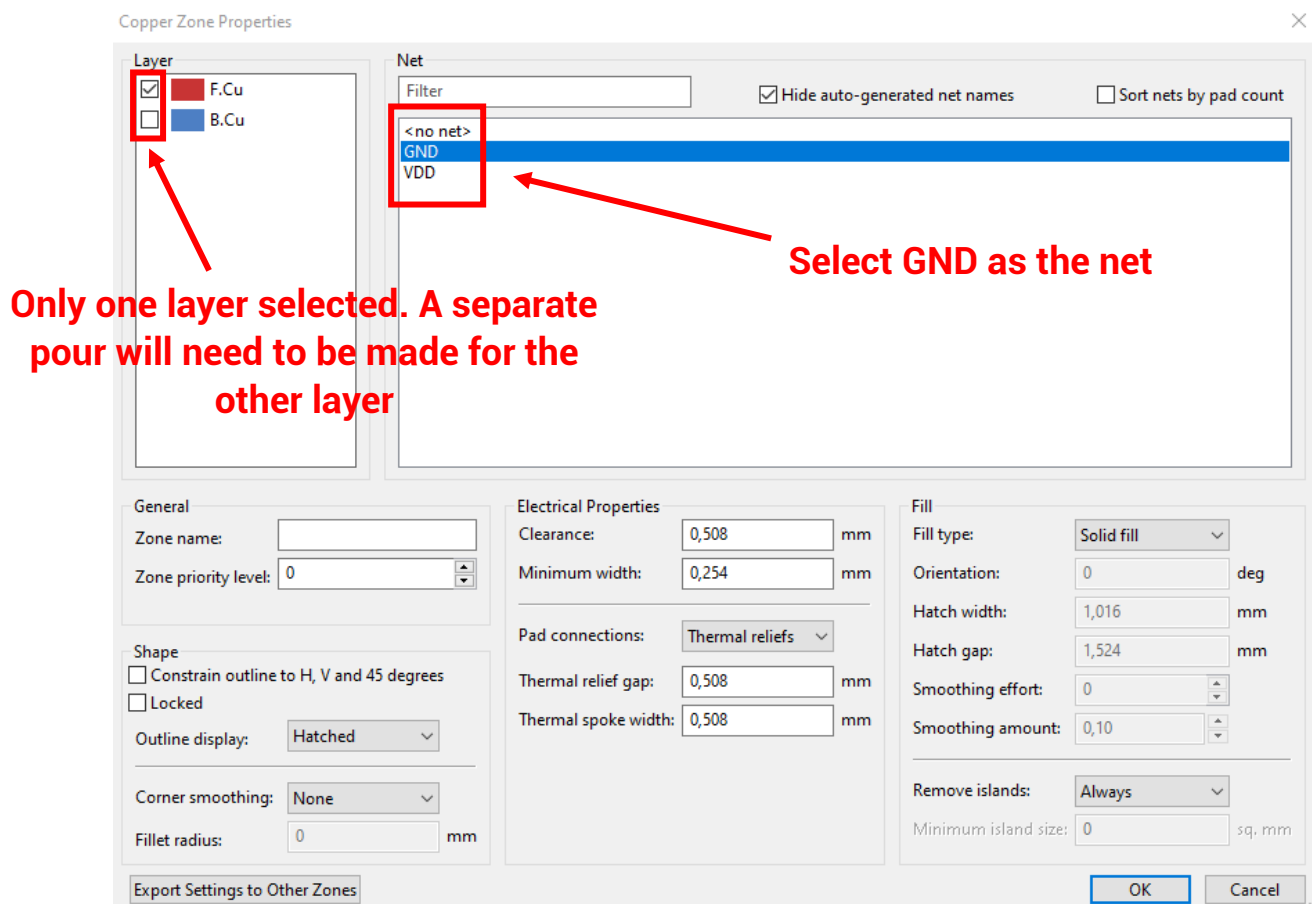
17. After GND is selected as the net, trace the outline of the board until it is complete. To view the poured zone, click on the "Show filled areas of zones" button on the left-hand toolbar. Then press the 'B' key to refresh the filled zone. (Notice how the pour does not intersect with any of the routes you placed. If you change routes later, but do not want to completely redo the polygon pour, simply press 'B' and it should correct itself!)

GND nodes auto-connect to the pour
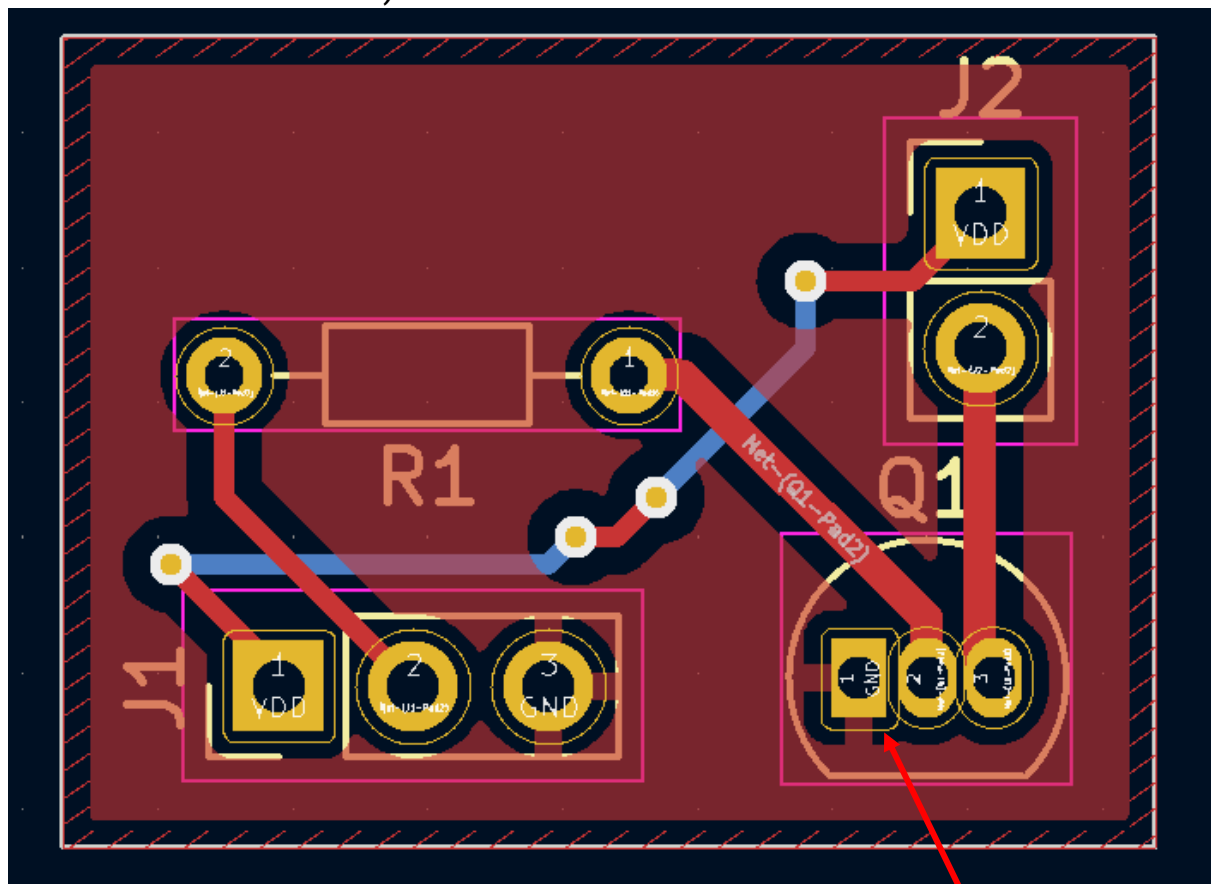
18. Do the same polygon pour for the B.Cu layer.
19. The PCB should finally be complete! You can even view it in the 3D Viewer under the 'View' tab on the top toolbar.
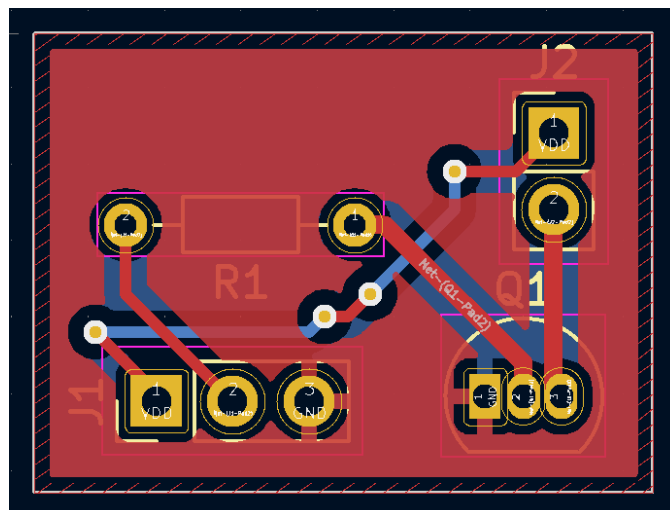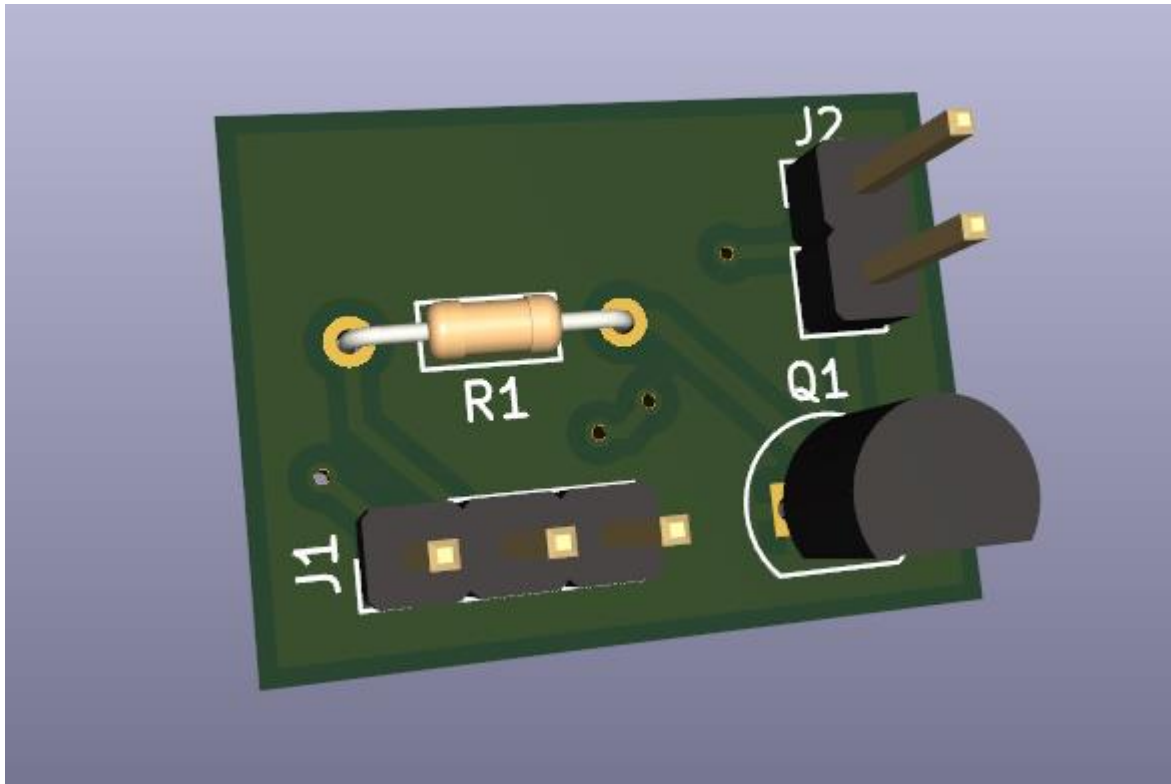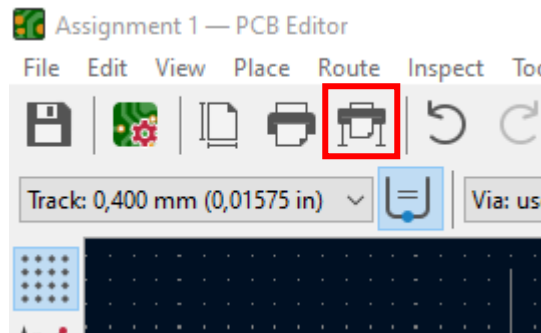
20. Unfortunately, manufacturers will not be able to print PCB's from KiCAD files. In order to convert it into a file format suitable for manufacturers, we need to generate gerber and drill files…

Part 3: Gerbers and Drill Files

1. To generate our gerbers and drill files, select the "Plot" button on the top toolbar.


2. The assignment brief should specify which layers to generate gerber files for, but generally, these layers are:
   - F.Cu
   - B.Cu
   - F.Silkscreen
   - B.Silkscreen
   - F.Mask
   - B.Mask
   - Edge.Cuts

   Make sure ONLY these layers are selected for the gerber files.

Make sure ONLY these 7 layers are selected

Make sure this is NOT ticked
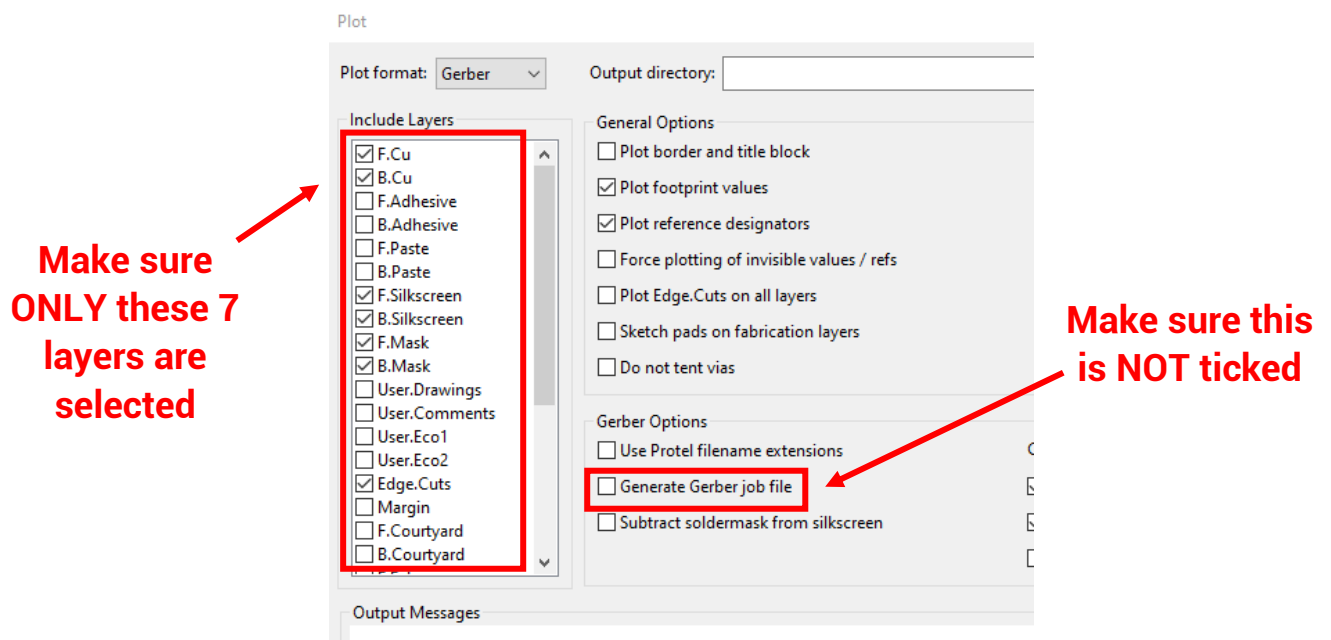
3. We need to assign a folder for all our gerber files to be saved in. Click on the Folder button next to the Output directory. Create a new folder called 'gerbers' within the existing project file. Select this folder.
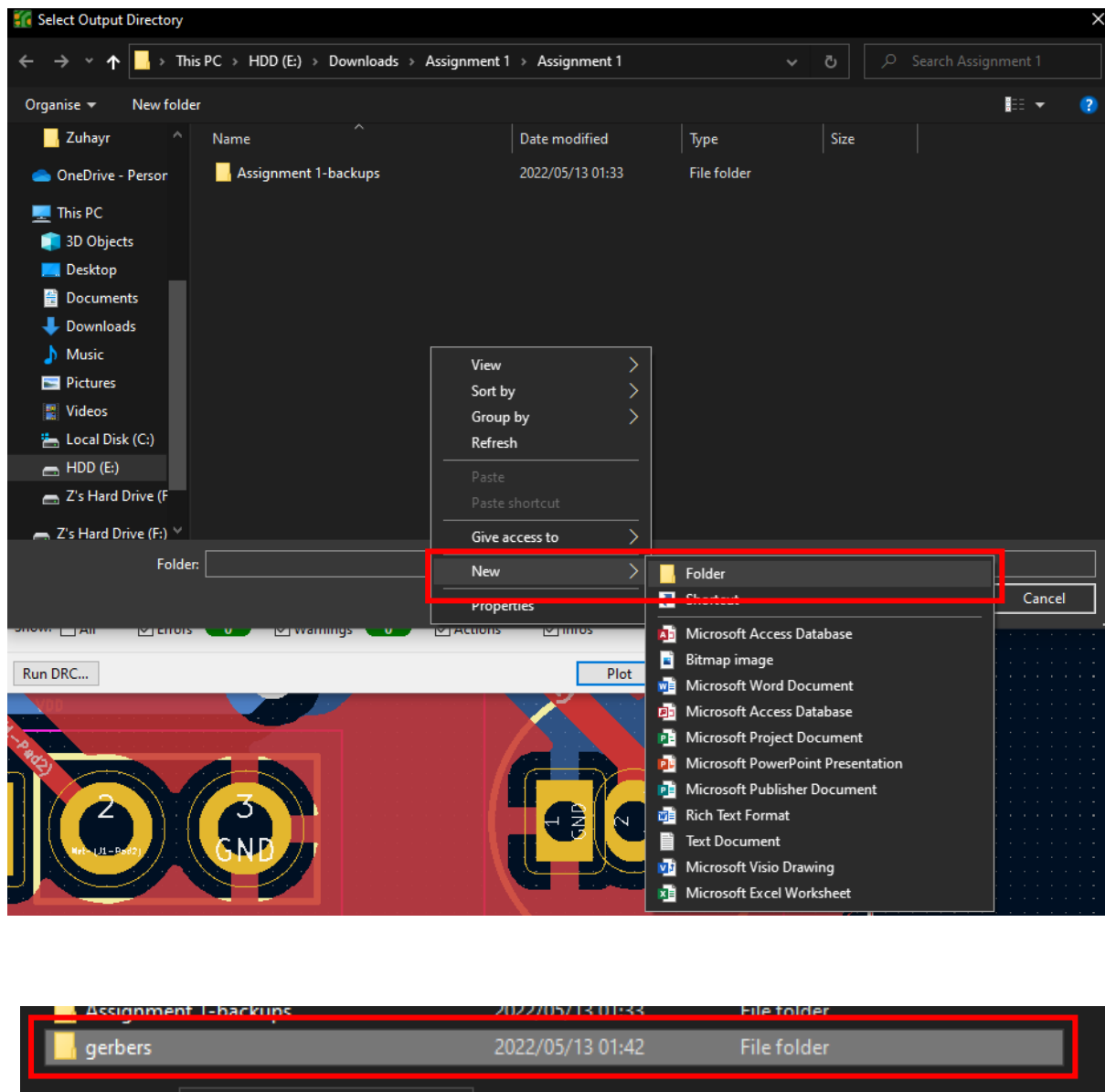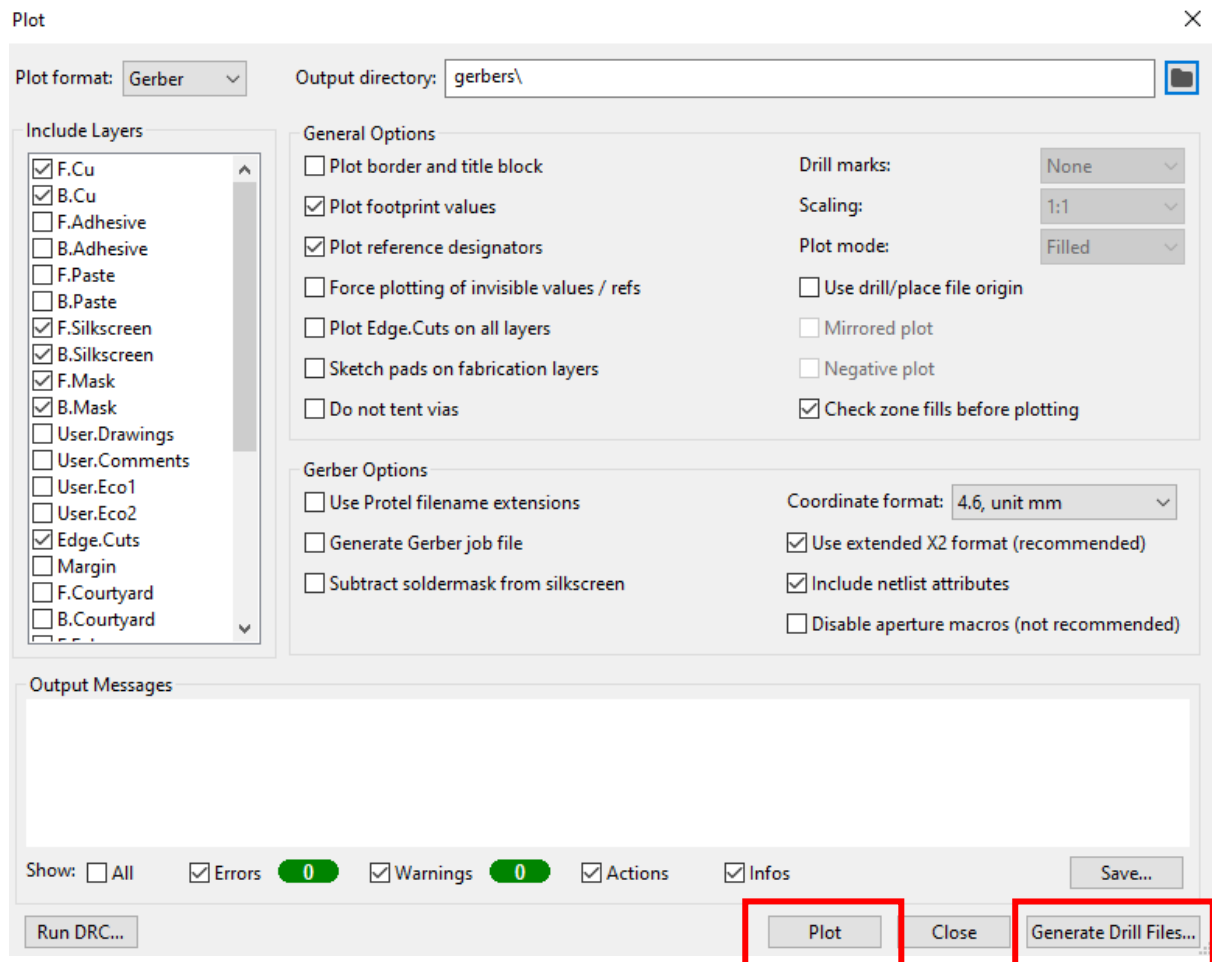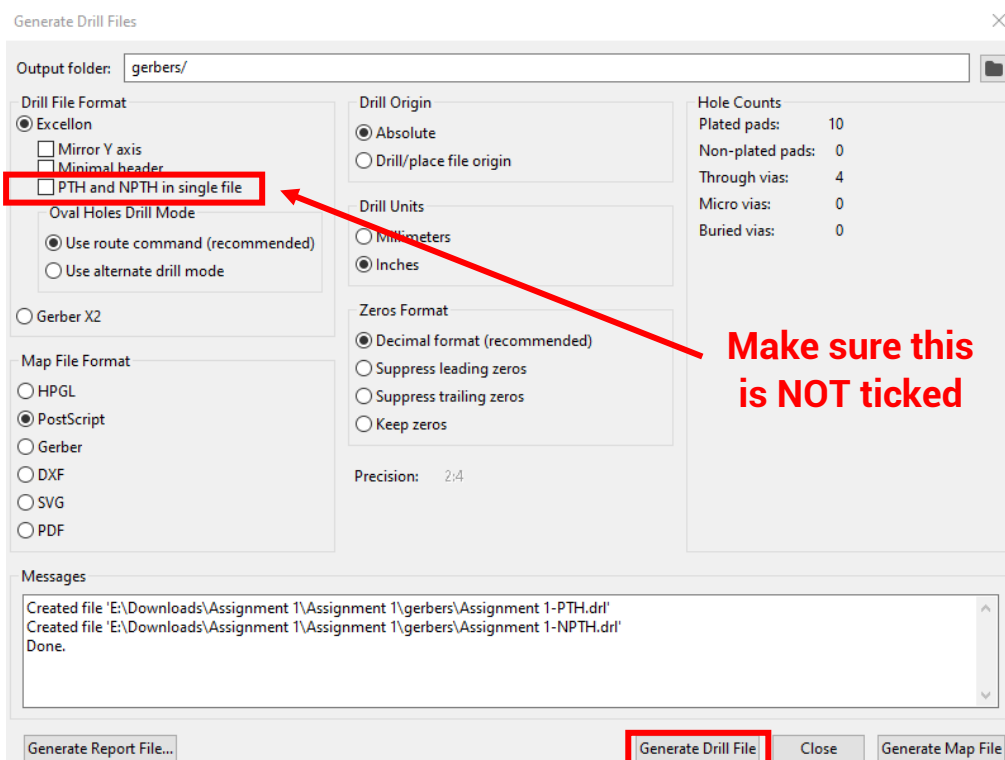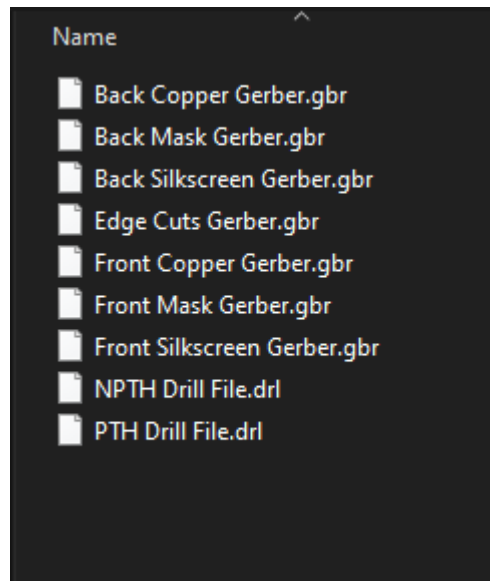
4. Click on the "Plot" button. The gerber files should be generated.

5. To get our drill files, click on "Generate Drill Files…". Make sure the "PTH and NPTH…" box is unticked, and generate your drill files.

6. You should now have 9 files in the "gerbers" folder: 7 gerber files and 2 drill files. Make sure to rename these files clearly and appropriately.

Name

📄 Back Copper Gerber.gbr
📄 Back Mask Gerber.gbr
📄 Back Silkscreen Gerber.gbr
📄 Edge Cuts Gerber.gbr
📄 Front Copper Gerber.gbr
📄 Front Mask Gerber.gbr
📄 Front Silkscreen Gerber.gbr
📄 NPTH Drill File.drl
📄 PTH Drill File.drl

7. That should be all we need to do for the assignment! (If you want to verify that your gerbers are correct, you can open them in the Gerber Viewer, but it is not necessary)

This is all I know for the assignment, but they could always throw a curveball or two in there. I will send the links for 2 very useful videos on our EEE group.

Really hope this helps. Good luck everyone!