UOS EEE Society KiCAD Lectures
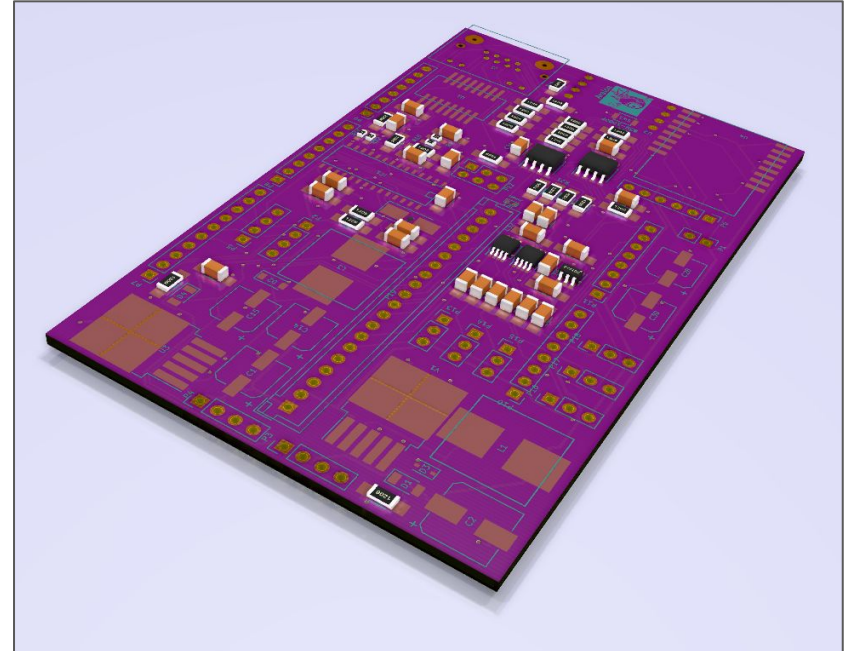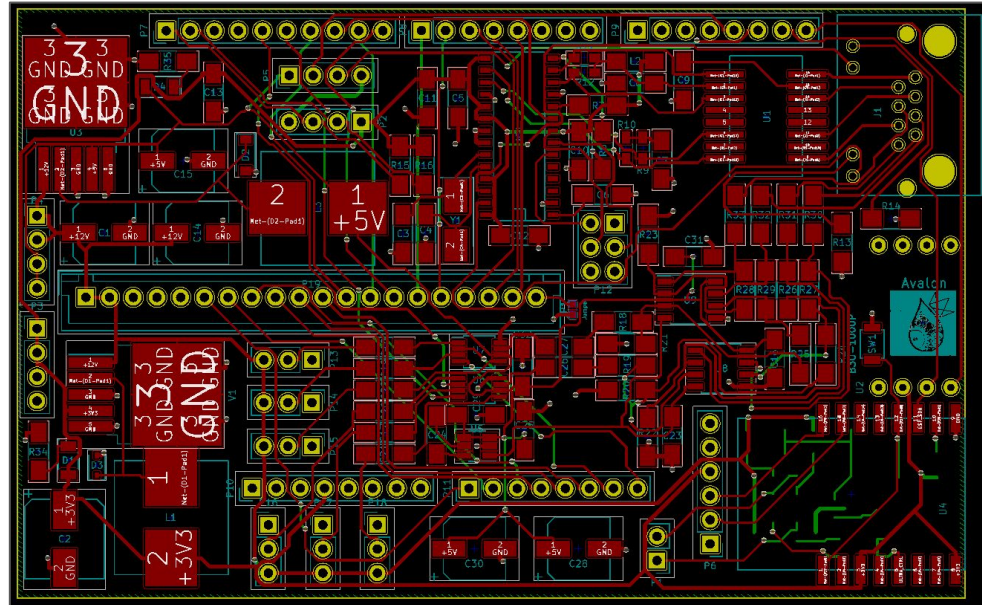
PCB Layout

Contents

1. What is PCB Layout?
2. The key steps to PCB Layout
   a. Draw PCB external dimensions
   b. Read Netlist
   c. Place Component Footprints
   d. Route Traces
   e. Fill Zones or Planes
   f. Amend Board Silkscreen
   g. Add Mounting Holes
   h. Generate Gerber Files
3. Seed Studio PCB Order Example
4. Extreme Design Cases

What is PCB Layout?

PCB Layout is the physical process of drawing (routing) the connections (traces) between the components of the circuit.

The key steps to PCB Layout

1) Draw PCB external dimensions
2) Read Netlist
3) Place Component Footprints
4) Route Traces
5) Fill Zones or Planes
6) Amend Board Silkscreen
7) Add Mounting Holes
8) Generate Gerber Files

This guide only explains two layer boards in detail. Multilayer boards will be briefly discussed.

Draw PCB external dimensions

Draw the external dimensions of the PCB:

- Must be done on the "Edge Cuts" layer.
- This shows the outer edges of the PCB, where it will be cut out of the raw FR4 material.
- You must ensure this is dimensionally correct, you can use the **Grid** or **Dimension Tool** to help you.
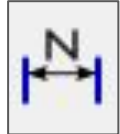
Draw PCB external dimensions

- The outer dimensions of the PCB are usually defined by your specification.
- If you have the luxury of unlimited space, then you can draw the external dimensions after you have placed and routed everything.
- External dimensions have to be regularly modified to make space for component changes or larger parts.

Read Netlist

- The Netlist is a large file that contains information on all of the connections between the components on the board.
- The Netlist is generated from the schematic. The "generate netlist" button is found on the top toolbar.
- An error in your schematic translates to an error in your Netlist ⇒ **BE CAREFUL!**

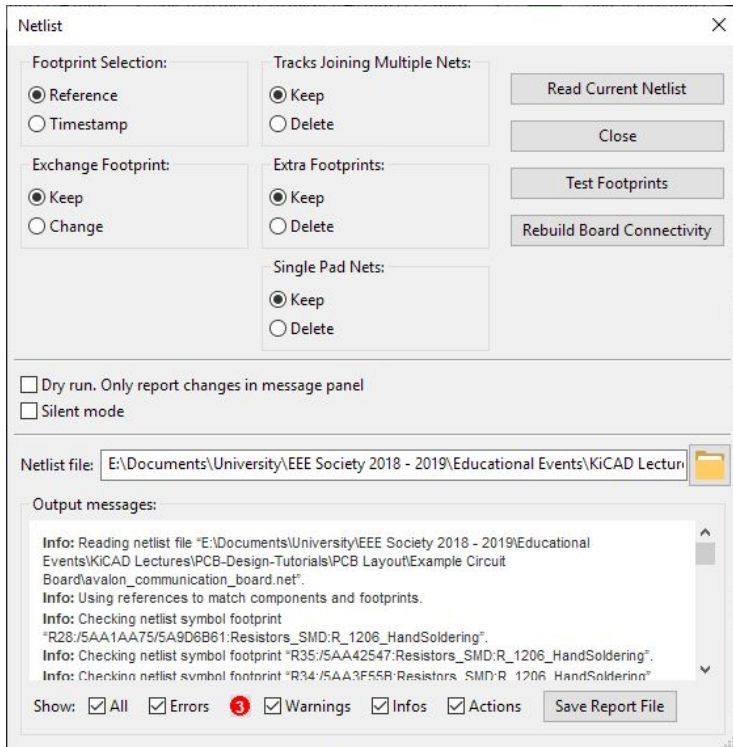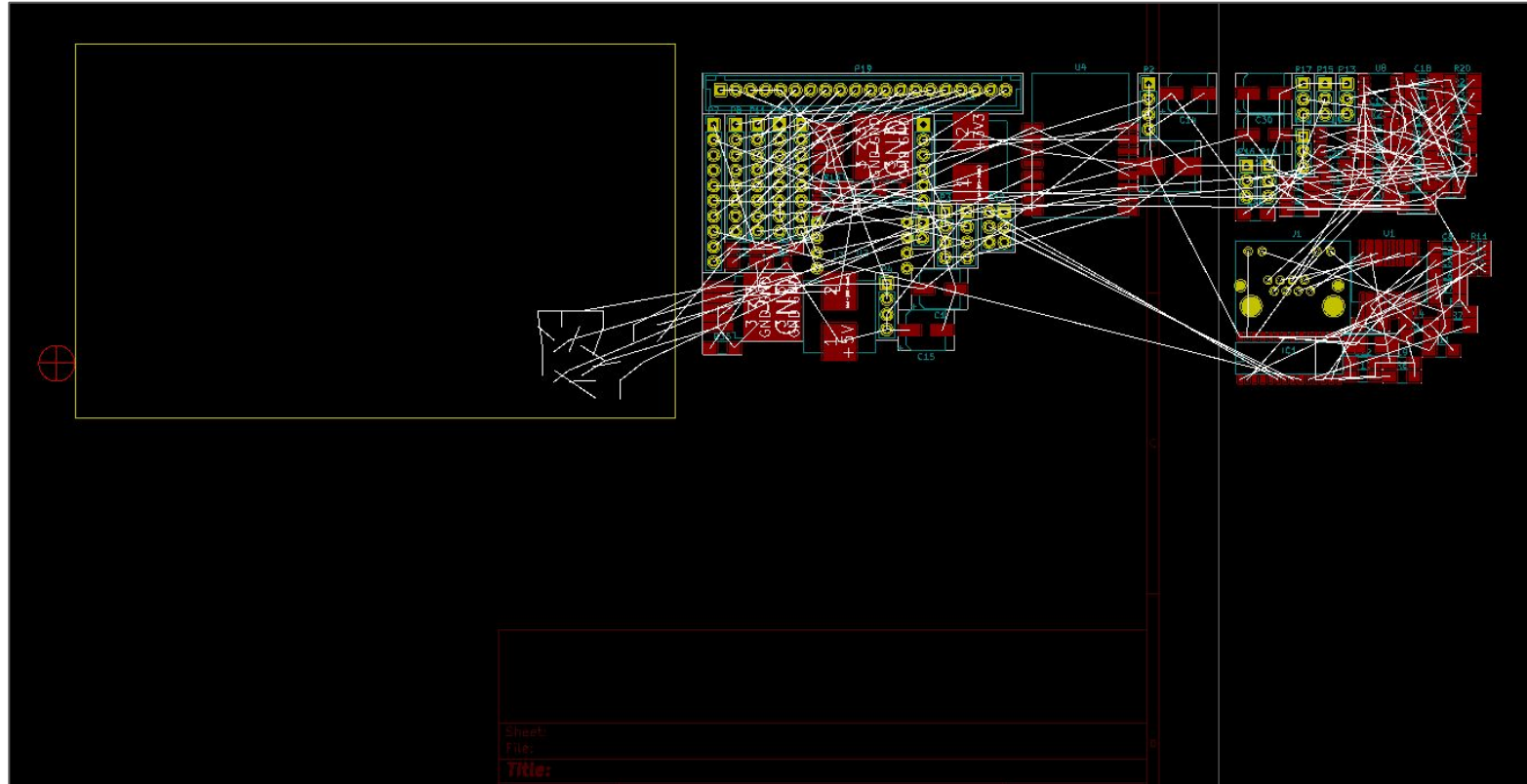**The "generate netlist" button in the Schematic Editor ⇒**

Read Netlist

Read the Netlist to transfer all component footprints to your PCB design:

- Click the "read netlist" button at the top of the editor.
- Click "read current netlist".
- The netlist window will inform you of any errors or missing footprints in the "output messages" window.
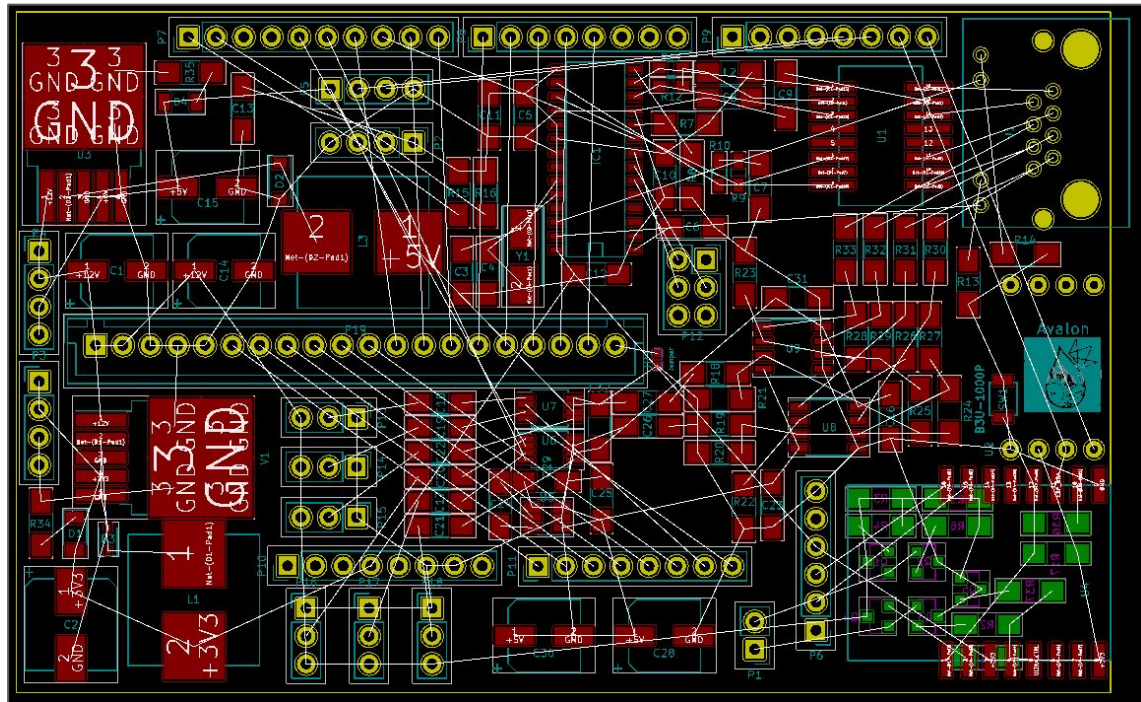
Read Netlist

Place Component Footprints

Place the components within the external dimensions:

- Ensure you move to the "Top Layer" (F.cu) or "Bottom Layer" (B.cu).
- Move components using the "m" keyboard shortcut.
- Place larger components first and then fit smaller components around them.
- Try to compartmentalize sections of your design.

Place Component Footprints

● Prioritise components that need to be close to the edge of the circuit board (connectors, antennas, etc.).
● Consider the heat components will dissipate, will sections of your current layout get particularly hot?
● Try and make your traces as short as possible by grouping components.

Usually Trial, Error and Intuition

Route Traces

Route the traces between the components on the board:

- The white lines are the "Rats Nest", they show what pads should be connected.
- Traces are placed using the "route tracks" button.
- Whilst using the route tracks button, keyboard shortcut "v" can be used to place a "via" to move between layers.
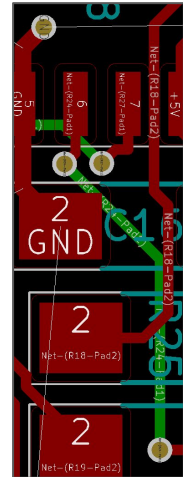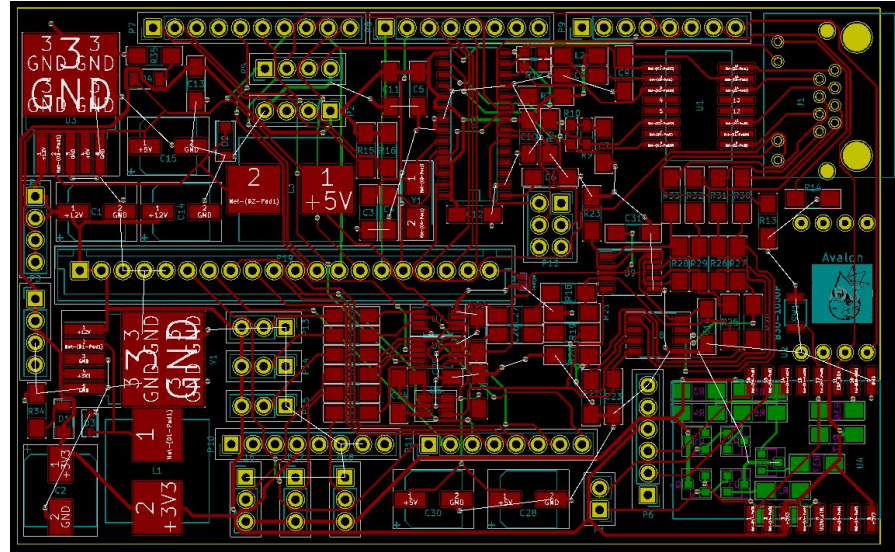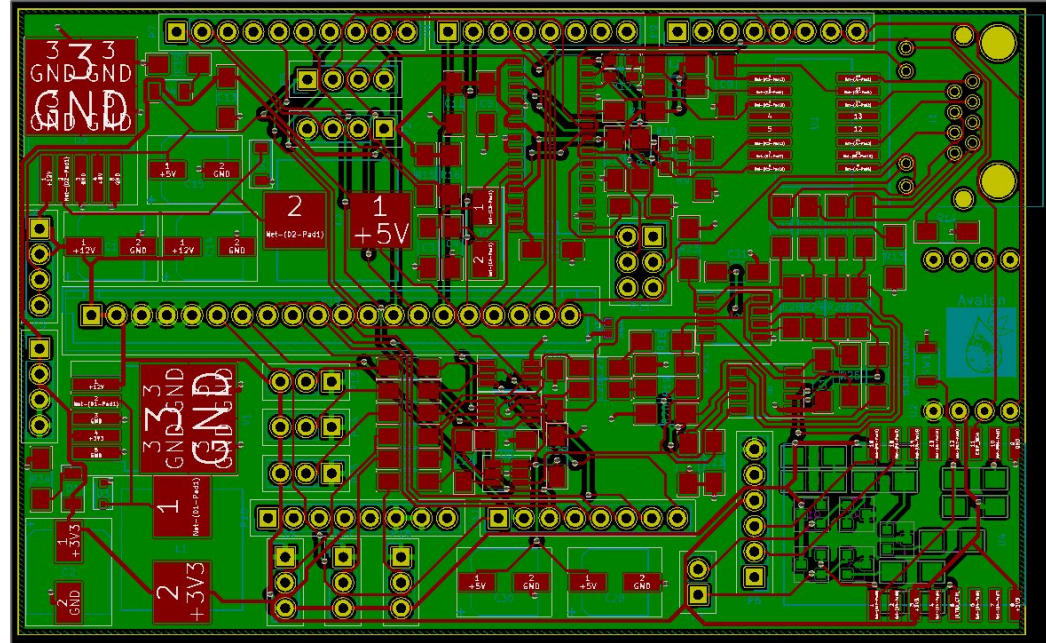
Fill Zones or Planes

Fill Power or Ground Planes to complete connections on the circuit board:

- The "add filled zones" button can be used to place zones.
- The "add keepout areas" button can be used to place areas not to be filled.
- Buttons on the left hand toolbar can be used to show or hide the zone colour.

Fill Zones or Planes

● Zones and planes are generally used for Power or Ground rails. They allow easy distribution of a connection that lots of components need.

● Generally for two layer boards, the back side is used as a Ground Plane.

● Multilayer boards can incorporate planes for all of the different Voltage Rails.

Amend board Silkscreen

Amend the board silkscreen to ensure all components are correctly identified:

- Move silkscreen text around to ensure that components are correctly labelled for soldering.
- Additional text can be placed on the board using the "graphic text" tool.
- Ensure you are on the "F.silkS" or "B.silkS" layers whilst doing this.
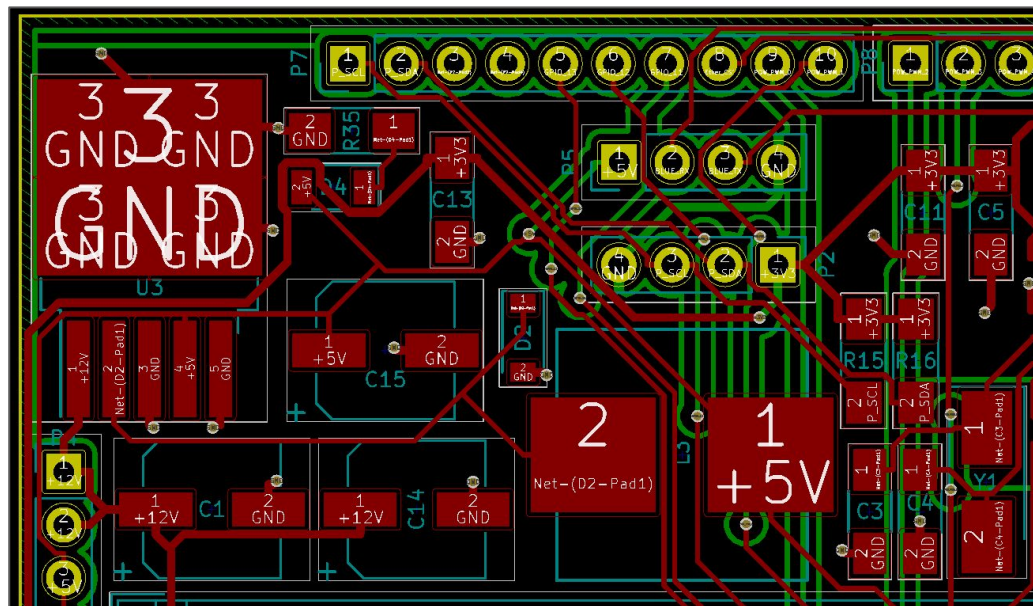
Add Mounting Holes

Mounting holes can be added to ensure that your board can be secured to a casing:

- The "add footprint" button can be used to place additional footprints on the board.
- Mounting holes can be found in the "Mounting_Holes" library.
- Mounting holes are regularly connected to the ground plane.
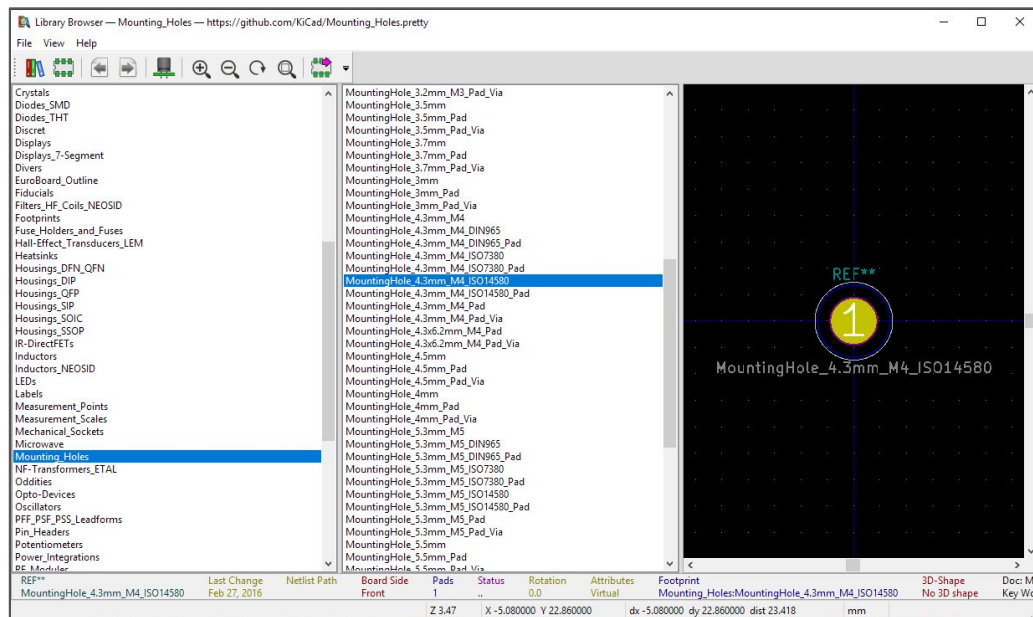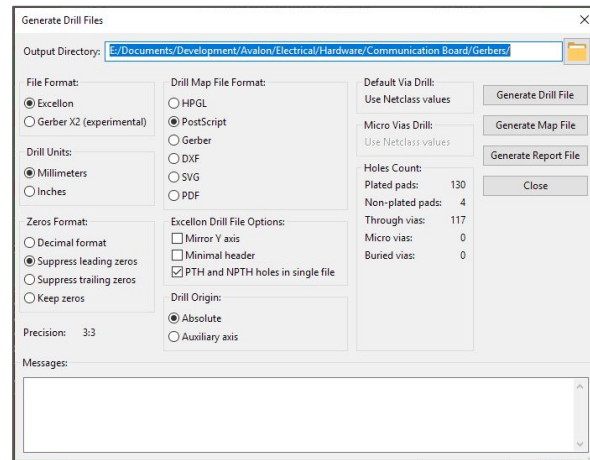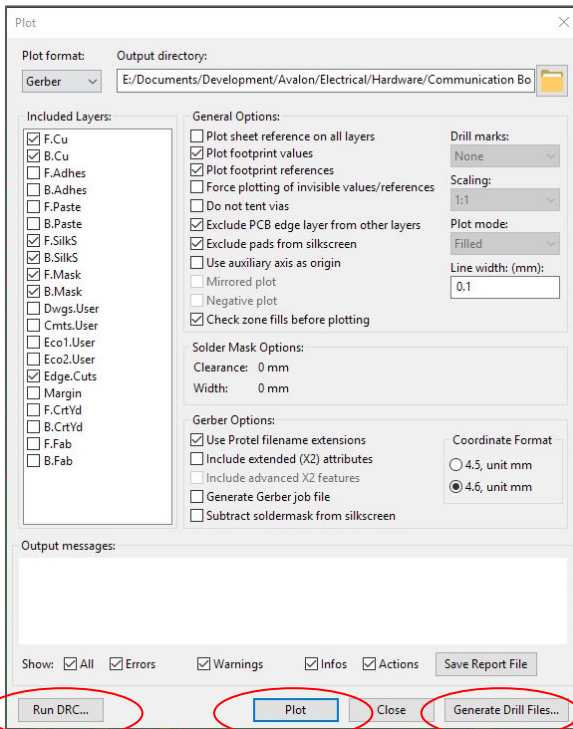
Generate Gerber Files

Generate Gerber Files to be sent to the PCB manufacturing house:

- Use **File ⇒ Plot** to generate the Gerber Files.
- Specific settings for the PCB house you are using will be provided on their website.
- Most manufacturers will have a guide on their website.
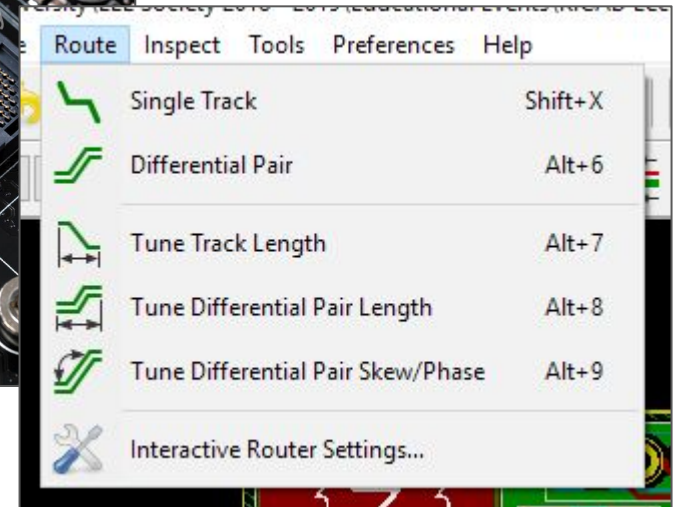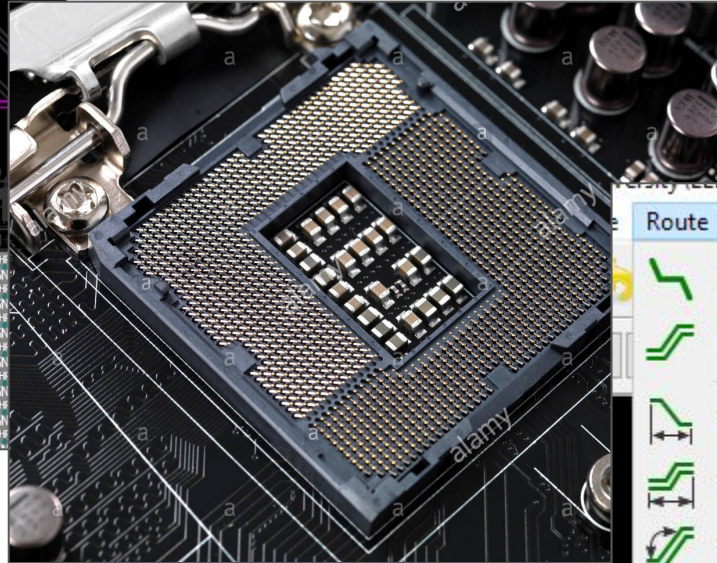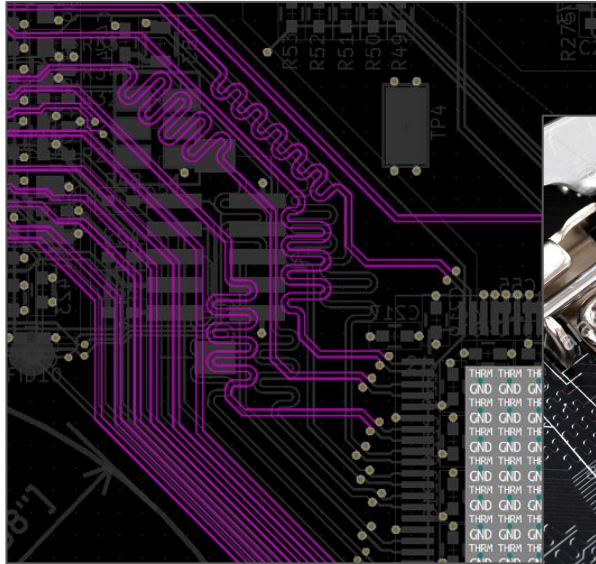- Remember to generate "Drill Files"!

SeedStudio KiCAD Guide

Seed Studio PCB Order Example

Extreme Design Cases - Differential Routing

Extreme Design Cases - Antennas

- KiCAD can be used to generate PCB Antennas.
- Recommended practice is to generate the Antenna as a custom footprint and then insert this into your PCB design.


KiCAD Antenna Design Guide

Extreme Design Cases - Multilayer Boards

- Multilayer boards become paramount when producing extremely complex boards. Modern computer motherboards can have 8 layers or more.
- **Setup ⇒ Layers Setup** can be used to amend the number of layers the board has.
- Whilst drawing traces, right click and press "select layer and place through via" to place a via to any of the boards layers.
- Sometimes it is useful to place standalone vias to reduce the resistance between areas of the ground plane.
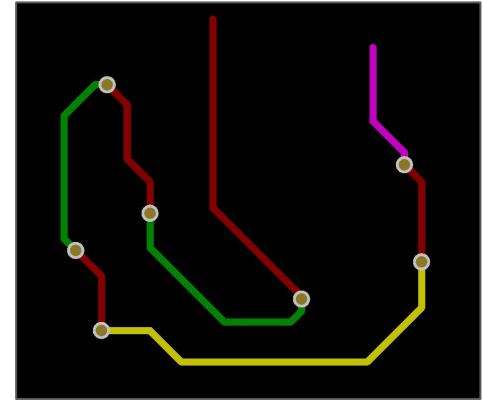
Summary

- PCB Layout is the process of placing and connecting components using traces or tracks.
- The key steps to PCB Layout are:
    - Draw PCB external dimensions
    - Read Netlist
    - Place Component Footprints
    - Route Traces
    - Fill Zones or Planes
    - Amend Board Silkscreen
    - Add Mounting Holes
    - Generate Gerber Files
- Extreme Design Cases ⇒ Differential Routing, Antennas, Multilayer Boards

Thanks for listening!

This is the last lecture in the series!

Follow us on Social Media:

- https://www.facebook.com/uoseeesoc
- https://twitter.com/uoseeesoc
- Snapchat - uoseeesoc
- Gmail - eeesoc@sheffield.ac.uk