EEEE2046 Energy Project

Power Supply PCB Design Task

1. PCB Design Task

You are required to design a PCB layout for your power supply circuit. This must be completed in the KiCAD package using the template project on Moodle as a starting point.

If you are new to KiCAD or PCB design (as expected for the majority of students), then please read through this document carefully as it details everything you need to know to complete the assessment.
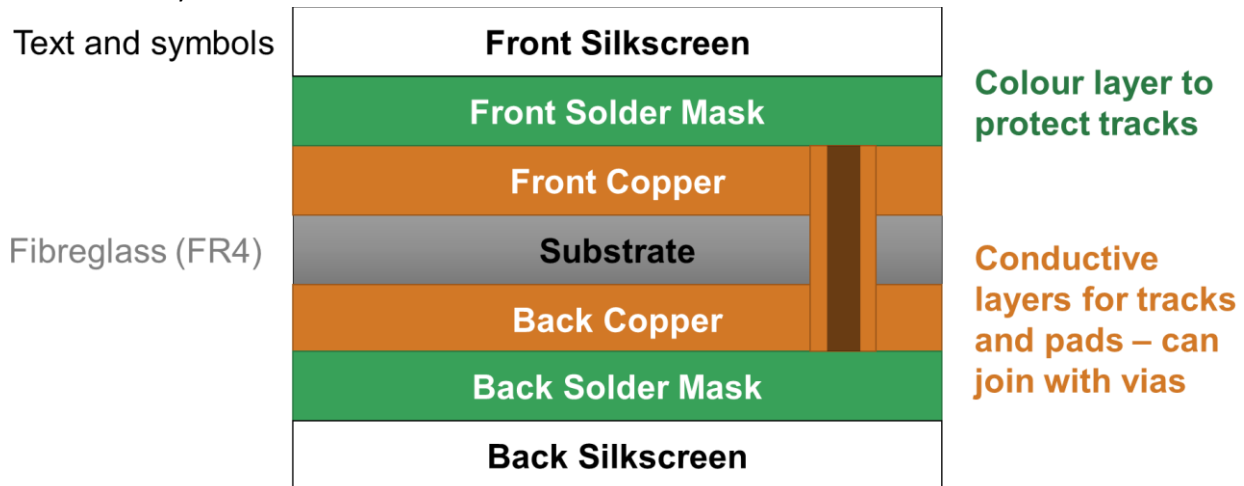
The provided KiCAD template project provides a completed main converter schematic, this is the circuit you are expected to produce a PCB layout for. Footprint assignments have been completed as therefore you can start directly from the PCB layout without making any further changes to the schematic.
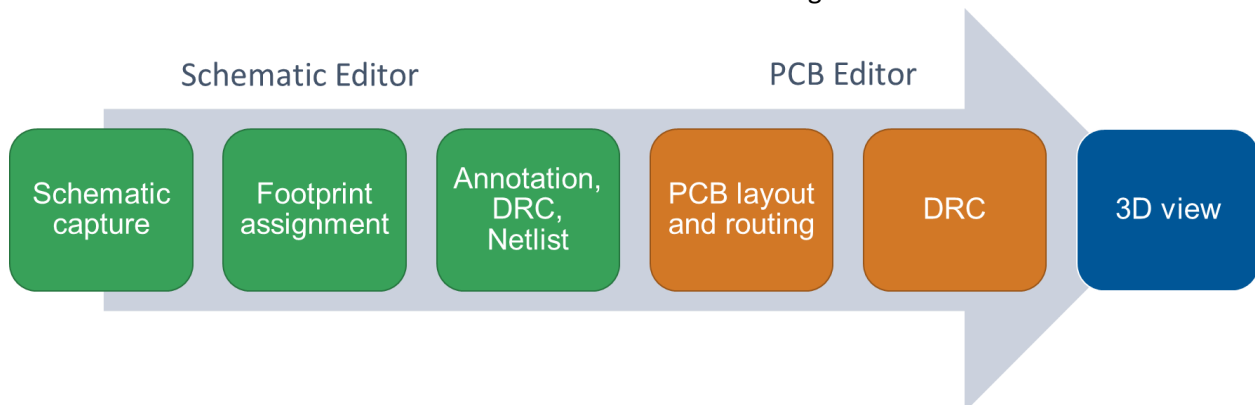
2. Introduction to PCB design

A printed circuit board (PCB) is a custom designed and manufactured sheet that uses copper tracks to connect components. The main benefits of using a PCB over other circuit construction methods are that it is fast to assemble (and this can be automated), allows for complex designs using multiple layers and unique component footprints, and provides mechanical support for delicate or large components. Unfortunately, PCBs can be expensive to manufacture, particularly in small quantities, and required specialist skills and software to design.

PCBs are constructed from a layered board, and layout design involves drawing and placing features on each of these layers:

Text and symbols	Front Silkscreen	Colour layer to protect tracks
	Front Solder Mask	
	Front Copper	
Fibreglass (FR4)	Substrate	Conductive layers for tracks and pads – can join with vias
	Back Copper	
	Back Solder Mask	
	Back Silkscreen	

Whilst each PCB design software has its own interface, most follow a common process to design a PCB involving tools for drawing a schematic of the circuit and then laying out the physical component footprints. The software can carry out automated checks (called Design Rule Check [DRC] in KiCAD) to confirm the schematic connections match that of the final PCB design.

Schematic Editor → PCB Editor

Schematic capture → Footprint assignment → Annotation, DRC, Netlist → PCB layout and routing → DRC → 3D view

3. Introduction to KiCAD

Modern PCB layouts are achieved using computer aided design (CAD) software. There are many software packages available but for this module we will be using KiCAD[1]. Using a CAD package for PCB design offers many advantages including:

- Reusable component libraries
- Linked PCB and schematic to prevent different connections between the two
- Automatic error checking for manufacturing limitations (e.g. track width and clearance)
- Quick creation of Gerber files required by the PCB manufacturer

4. Install and run

KiCAD can be downloaded from https://www.kicad.org/download/
There are packages available for all major operating systems and these can be installed without a licence. You can also find KiCAD on the engineering virtual desktop.

A template project has been provided on Moodle. It is important that you use this because:

- It includes a component library with schematic symbols and footprints for the converter components you will be using.
- It provides a schematic of the converter circuit so that you can start from the PCB layout part of the design process.
- The PCB design contains edge lines to give a board dimension of 9x4", which you must keep within[2]. Mounting holes are also provided at each corner to support your converter.

Download the template project from Moodle and extract it into a suitable directory on your PC.

Launch KiCAD and select 'File'→'Open Project'. Select the template project file (EEEE2046_Template.pro) and click 'open'.

---

[1] KiCAD has been chosen as it has cross-platform support (Windows, Linux and Mac) and is free to download and use. This means you can install it on your personal computer without worrying about licencing.
[2] Inches has been used to better match the pin spacing of through-hole components. The copper layers must fit within the given edges.

5. Drawing your schematic

This section (along with 6 and 7) are provided for reference and should you wish to make any additions/changes to the schematic (for example the addition of test points).

Load the schematic editor by clicking on:

You can add components to your schematic by clicking on on the right toolbar or by using the hotkey 'a', followed by clicking anywhere on the schematic page. The search tool is the easiest way to find component symbols some examples include:

- 'r' for resistor
- 'c' for a non-polar capacitor
- 'cp' for a polarised capacitor
- 'd_zener' for a Zener diode

Components specific to the project are included in the 'EEEE2046_electrical' library. These include:

- BYG20J
- BYV29FX-600
- ECA2AM102
- ECA2AM471
- ETD34_L
- ETD34_Transformer_1P_1S
- IRFI530NPbF
- MBRB3030CTLG
- STPS20H100CFP
- TC4425AVPA
- TO220_Heatsink
- Toroid-Transformer_1P_2S
- VS-KBPC606PBF

Power symbols can be added using the button or the 'p' hotkey. Every power symbol of the same type is connected in the schematic. This allows you to keep the schematic tidy and readable.
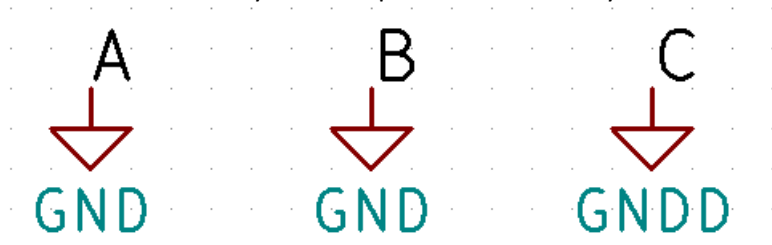

Figure 1 - Power symbols. A and B are connected because they connect to the same symbol type. C is separate.

There is a special power symbol called 'PWR_FLAG'. This is used to tell the error checker that connected nodes are powered externally. It is good practice to connect any external power symbols to a PWR_FLAG in your schematic.
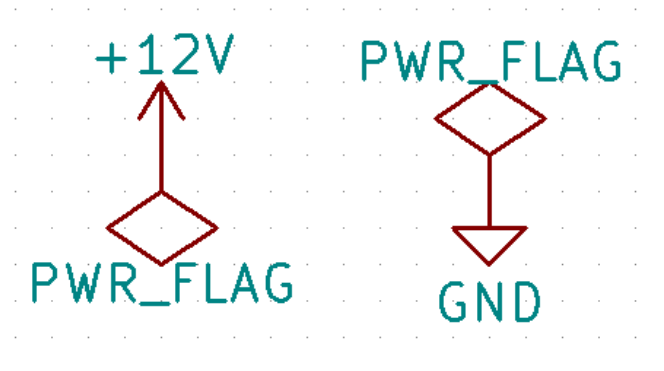
Figure 2 - PWR_FLAG symbol used to indicate that +12V and GND are provided by an external power source.

Connections between components can be created using the wire tool [icon] or the hotkey 'w'. Click once to start a wire, additional clicks will create corners, unless on a component terminal where they will end the wire. A double click will end the wire immediately. Wires will always be drawn as straight horizontal or vertical lines.

Connections can be named using the net name tool, [icon] or the hotkey 'l'. Connections with the same name are connected. These labels can be used to make your schematic neat and easier to read.

The not connected flag [icon], hotkey 'q' can be placed on unused component pins. This prevents the error checker from giving a warning.

General important hotkeys are:
- 'm' to move components (breaking connections)
- 'r' to rotate components by 90deg (can be used while moving a component)
- 'g' to grab and move a component (keeping connections)
- 'c' to copy a component
- 'e' to edit a component and 'v' to just edit the value of the component
- 'esc' to cancel the current action
- 'del' to delete components, wires, symbols, etc
- The mouse wheel can be used to zoom and middle-click can be held to pan

For a neat layout you might want to consider using the line tool [icon] and the text tool [icon]. These allow you to visibly break your schematic into sections and add key information and comments.

6. Assigning component footprints

Before we can create a PCB layout, we need to assign each component a footprint. This is achieved by using the hotkey 'f' over a component symbol or by right-clicking the component and going 'Edit Component' → 'Footprint'.

Click 'Select' in the window that opens to get the library browser from which a suitable component footprint can be selected from the available libraries.

The components in the EEEE2046_electrical library have been pre-assigned footprints. Components from the standard libraries will need to be assigned a suitable footprint. Make sure to check the dimensions of the components you will be using and ask for help if you are unsure.

For example for a standard through-hole resistor the:

'R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Horizontal' component from the 'Resistor_THT' library is a good choice.

7. Annotation and ERC

Component symbols must be annotated so they each have a unique name. This can be done automatically by the annotate tool in the top toolbar .

Electrical Rule Checking (ERC) can be performed by clicking the button. This will notify you of unconnected pins and other basic errors. It will **not** test the functionality of your circuit or that connections are correct.

8. Creating your PCB Layout

Start by opening the PCB editor by clicking , (this can also be done from the schematic editor).

The edges for your PCB and four mounting holes have already been placed. You will be designing a two layer board and can therefore route tracks on the Front (F.Cu) layer and Back (B.Cu) layer. In KiCAD these are colour coded to be red and blue respectively. You can also place text, component names, and other artwork on the silkscreen for these layers (F.SilkS, B.SilkS). This will be printed as white text on the board.

To load the components from our schematic into the PCB tool we need to update the layout. Click the button to open the import tool and click 'Update PCB'[3]. After closing the update window, you will be able to click place all the new footprints that have been imported.

Footprints can be moved using the 'm' hotkey and rotated with 'r' as in the schematic editor. The schematic connections between components are shown as white lines called the ratsnest. When laying a track connected pads are highlighted.

Start by laying out your components on the board so that distance between connections are minimised. You will need to consider:

- The size of additional parts around the components e.g. heatsinks. (A heatsink component has been provided in the EEEE2046_electrical library, this contains a silkscreen outline of the heatsink we will use).
- The inductance of the tracks you will need to make – some inductance is unavoidable but consider for which connections it will be critical. You can reduce inductance by using shorter, thicker tracks.
- What decoupling capacitance will be required and where it should be placed to be most effective.
- Clearance between components and connections.
- How external connections will be made to the board for testing – including power, load, and debugging. You might wish to revisit the schematic to add some additional test points.
- Labelling of components and connections.

Through-hole component pads appear on both sides of the board and electrically connect both layers (through-hole plating) whereas surface mount components have pads on only a single layer. A via can be placed to switch between layers using the 'v' hotkey. A component can be flipped onto the other layer using the 'f' hotkey. Think about the height of the components before doing this – will it make it difficult to test the board?

---

[3] There are options to control what gets overwritten in the PCB when it is updated. You may need to change some of these when importing an updated schematic part way through a design.
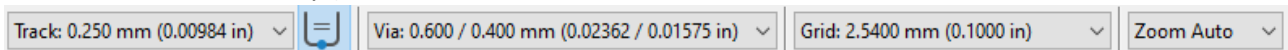
9. Routing tracks

With the components in place it's time to route tracks between them. Consider the current each track will be carrying – a high current will require a thicker track. For example, the default track width of 0.25mm can carry about 1.15A (with a 20C temperature rise in a 25C ambient, on an external PCB layer made from 1oz copper).

The calculator tools (accessible from the main KiCAD window, by the button) can be used to calculate the current capacity of different tracks.

Several track thickness and via sizes have been pre-configured for you to use in the template and these can be changed if needed in the PCB setup, , under the "Pre-defined Sizes" menu. You can select the track and via[4] you are using with the drop-downs at the top of the screen (the track width can be changed using the 'w' and 'ctrl+w' hotkeys):

| Track: 0.250 mm (0.00984 in) ∨ | Via: 0.600 / 0.400 mm (0.02362 / 0.01575 in) ∨ | Grid: 2.5400 mm (0.1000 in) ∨ | Zoom Auto ∨ |

After selecting the dimensions you want to use, start routing a track by clicking the button or pressing the 'x' hotkey. Click a pad to start routing that connection. The track will automatically route in straight lines at 45° angles and avoid other tracks and pads. When routing connected pads and tracks will be highlighted.
The hotkey 'v' can be used to place a via and change layer. When making a high-current connection between layers use large vias, lots of little vias, or a combination of the two.

10. Filled zones

Filled zones can be used as an alternative to tracks either to connect lots of pads or to make high current connections.

To create a filled zone click the button. Then click where you want the start of the zone to be. A menu will open letting you select the layer that the zone is on and the signal it connects to. After clicking 'Ok' you can map out the zone by placing corners and complete the zone by double clicking.

Visibility of filled zones can be toggled using the and buttons on the left toolbar. If you have routed tracks or moved components the filled zone can be refreshed by pressing 'b'.

11. Silkscreen layers

The silkscreen layers can be used to add key information to your PCB design. For example, component labels, identifying text, names of external connections, etc. As a starting point the template includes a text block that you **must** edit ('e' hotkey) with your name and date of your design. Feel free to move this as needed on the board.
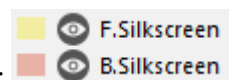
The silkscreen layers can be selected using the layer menu on the right toolbar: F.Silkscreen B.Silkscreen .

---

[4] A via is a small pad and hole on the PCB that joins the top and bottom layers.

You can add lines, shapes, and text using the tools on the right:

It is important to make sure your silkscreen is visible – if you have text over a component pin hole it will be drilled through!
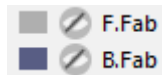
Switch off the visibility of the F.Fab and B.Fab layers[5] and check/move your silkscreen text so that it is suitability positioned. You will also get warnings of obscured silkscreen from the design rule check.

12. Design rule checking, unconnected pads and the 3D viewer

After completing your design, it is important to check that none of the design rules have been broken and that all the connections have been made.

Click to open the design rule checker then click 'Start DRC'. Any errors that have been found will be listed in the box and can be clicked to view on the design.

In the DRC window the 'List Unconnected' button can be used to show any pads that have not been connected as the schematic shows. Double clicking on a listed item will focus it in the PCB design window.

If you are unsure of any error messages or warnings in the DRC please ask.

The 3D viewer (Alt+3) can be used to create a rendering of your completed board design, including components. Check this carefully for any issues, e.g. overlapping components, obscured silkscreen and, hard to reach connectors, then make any required changes.

When you are happy with the design, and the DRC and Unconnected lists are empty, it is time to generate the Gerber files that will be sent to the manufacturer.

---

[5] This will hide most of the additional text and outlines for the components.

13. Gerber file generation and checking

This section is provided for completeness and your own reference. You do not need to produce gerber files for this assessment.

To generate the Gerber files that will be sent to the manufacture click on the plot tool 🖨. Setup the window as shown (setting the output directory to your initials):
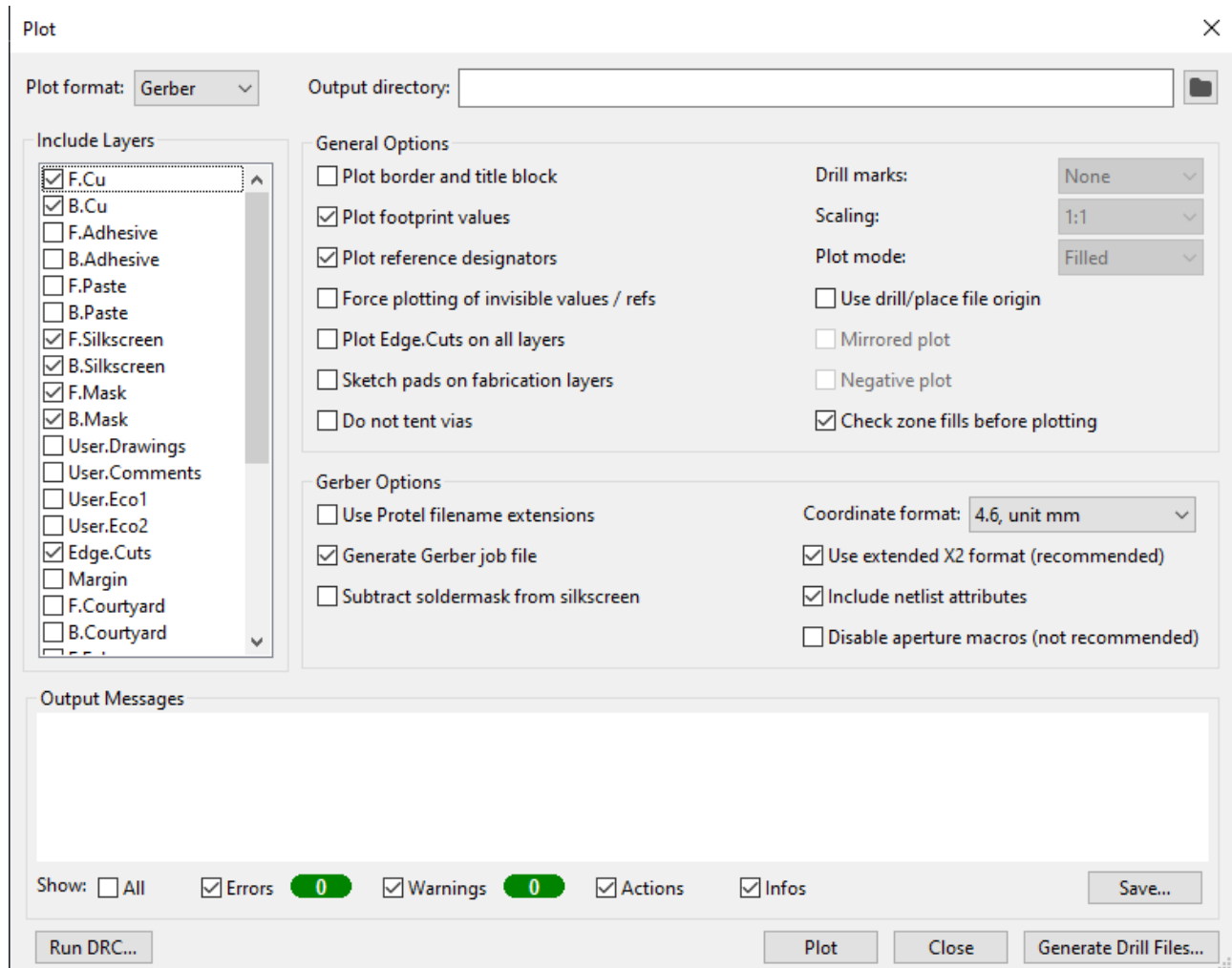
This will create a new folder in the project directory and save the Gerber files there. Click 'Plot' to create the files.

Click the 'Generate Drill File' button, then leave the settings unchanged and click the 'Drill File' button. You can now close both windows.

Save and close the PCB editor. Then open the Gerber viewer by clicking [icon].

Use [icon] to load individual layers – start with the Edge.Cuts followed by the copper layers. Check that the design is as expected.

Use [icon] to load the drill file check that the points are correctly sized for the pads and placed centrally.

If you have to make a change to the PCB design, then delete the gerber folder in your project directory before generating new files. This will prevent any old ones from mistakenly being used.

14. Submission

To submit your board design ZIP the whole project folder by clicking in the main KiCAD window and choosing a save location. Upload the ZIP file to Moodle.

Alongside this you should submit a PDF of one A4 side, detailing the design justification of your layout. This should answer the considerations given in the bullet points at the start of section 8.

15. Rubric

This assessment focus on both your ability to use the KiCAD software to produce a PCB layout, the quality of your submitted layout, and the valid justification of your design in your supporting PDF.

Use of KiCAD	
1	Only the most basic software functions have been demonstrated (component placement and track routing). Design itself might be incomplete or clearly non-functional.
2	Design demonstrates basic function use with some clear errors in use (e.g. undeleted track stubs). Design is likely function and passes DRC without errors.
3	Design demonstrates good use of the main software functions to produce a likely functional design, aspects such as supporting silkscreen text is missing.
4	PCB design demonstrates competent use of the software and use of functions relevant for the design of this circuit. Some functions might not have been used, but these do not affect the potential functionality of the design.
5	PCB design demonstrates use of all the main functions of design software including placement of components, routing of tracks, use of vias, filled zones, custom silkscreen text. The board passes the DRC without errors, and any warnings are explained in the supporting PDF.

Quality of layout	
1	Very poor and/or incomplete layout, may contain unconnected components and/or overlapping parts that would prevent function (or make construction/testing difficult).
2	PCB layout is mostly complete but there are significant errors visible in the design and/or captured by the DRC. No evidence of design considerations, relevant to power electronic circuits, have been implemented.
3	PCB layout is functional and passes DRC but unoptimized in many aspects. There is little to no design considerations made, relevant to power electronic circuits.
4	PCB is generally well laid out making good use of the board space. Some sections may not be optimal but there is evidence of design considerations relevant to a power electronic circuit.
5	PCB is well laid out with clear consideration made to component placement to minimise track length and ease connection/testing. Connection points are clearly labelled.

Justification	
1	Documentation provides very little specific detail and may fail to justify the design. The design may or may not reflect good practice or the points in the documentation.
2	Documentation explains basic aspects (e.g. track width choices) and these are reflected in the design. Deeper justification is missing or incomplete.
3	Documentation provides clear justification for the main design considerations, and this is reflected in the PCB design. There is some missing / unexplained design aspects that would be expected to have been included.
4	Documentation is mostly complete with good use of calculations/references where relevant. Design reflects the documentation.
5	Documentation justifies the design considerations, including calculations/references where necessary. The design reflects these justifications.