University of Nottingham
UK | CHINA | MALAYSIA

EEEE2046

Converter PCB Design

Energy Project

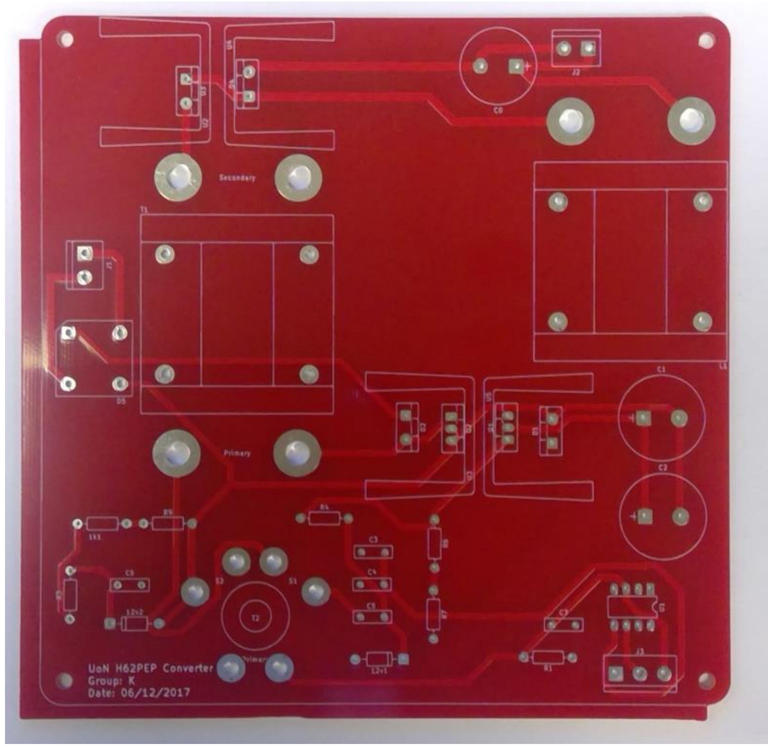
Introduction

- What is a PCB?
- PCB design process and software
- Key considerations for a power converter
  - General layout
  - Track width (and via size)
  - Inductance (and decoupling)

What is a PCB?

What is a PCB?

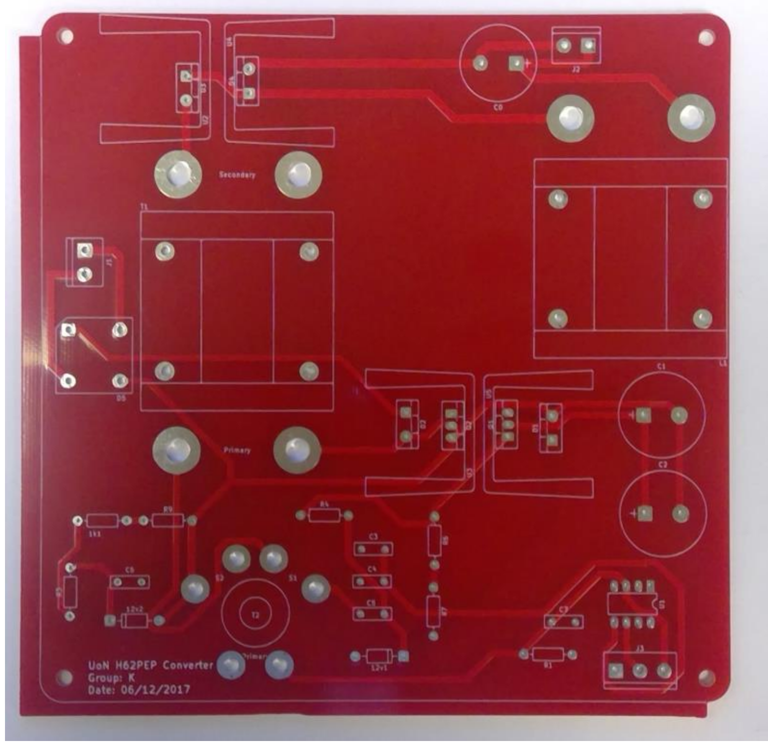
- **P**rinted **C**ircuit **B**oard
- Custom designed, manufactured sheet that contains copper tracks to electricaly connect components



Why use a PCB?

What is a PCB?

- **P**rinted **C**ircuit **B**oard
- Custom designed, manufactured sheet that contains copper tracks to electricaly connect components
- Benefits:
  - Faster to assemble – can be automated
  - Allows for complex designs – multiple conductive layers
  - Custom pad/hole spacing – SMD, switches, etc.
  - Mechanical support
- Drawbacks:
  - Expensive to manufacture – particularly in small quantity
  - Requires specialist design software and skills

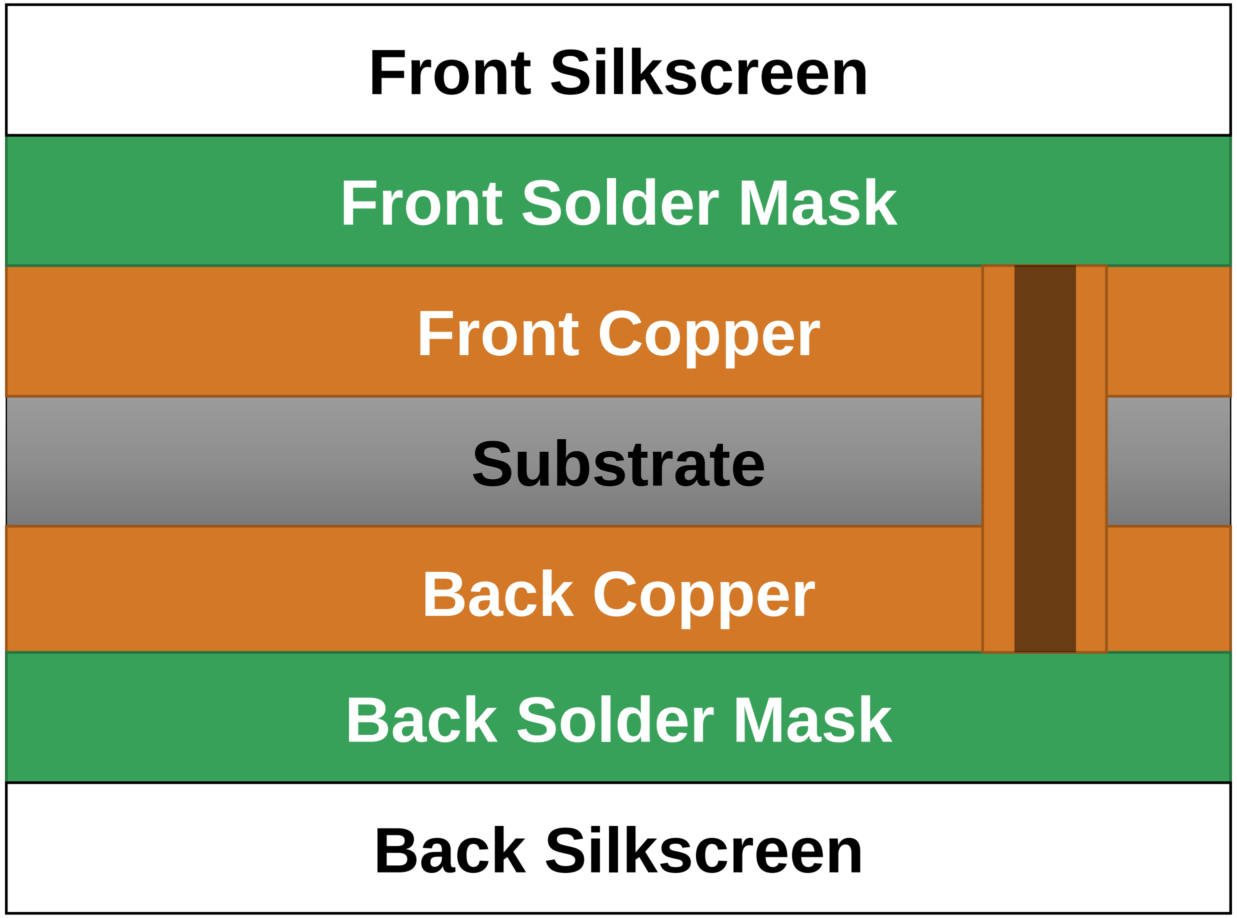


What is a PCB?

- **Layers – double sided board**

Text and symbols

Fibreglass (FR4)



Colour layer to protect tracks

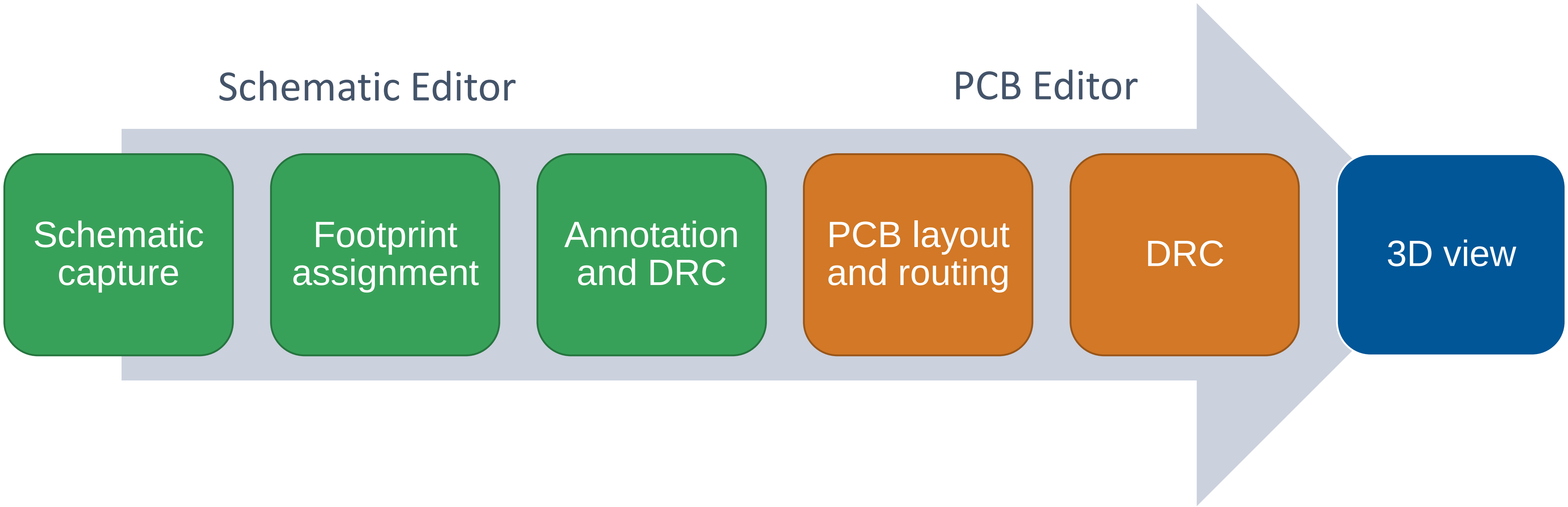
Conductive layers for tracks and pads – can join with vias

Design process

Software

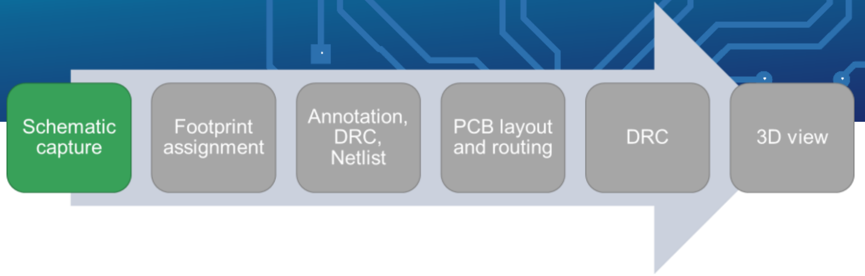
- Lots of packages available for PCB design
- We will use **KiCAD**
  - Free and open source – no licence required, no board limitations
  - Compatible with Windows, Linux, and Mac
  - Available at: http://kicad.org/download/
  - Pre-installed on the lab PCBs and in the tower computer room

PCB Design Process

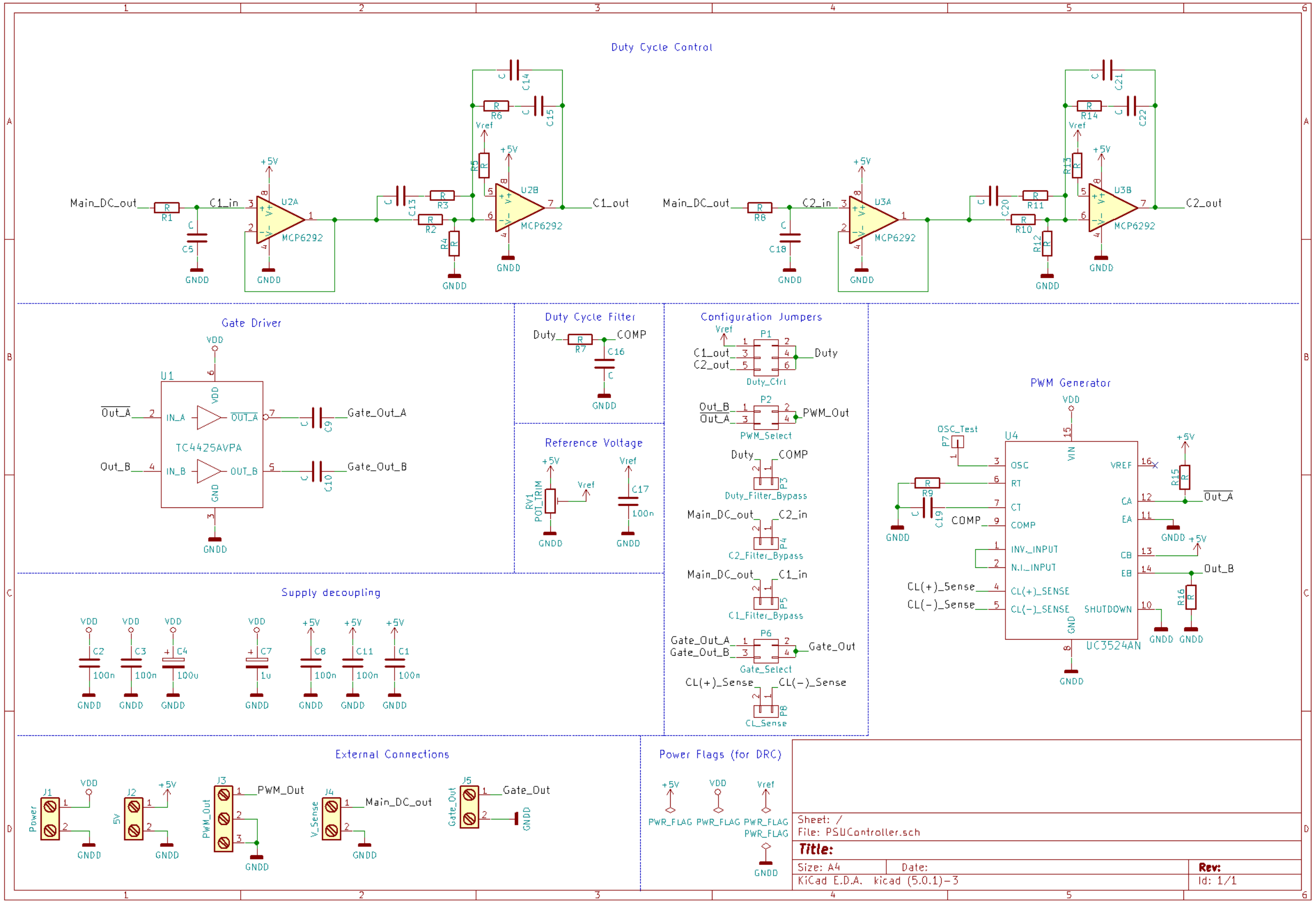


DRC = Design Rule Check

PCB Design Process



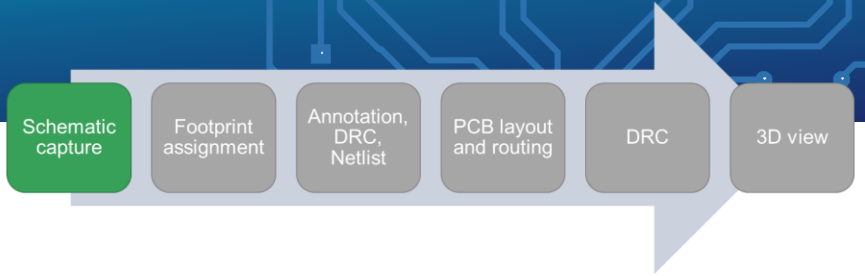
- **Schematic Capture:**



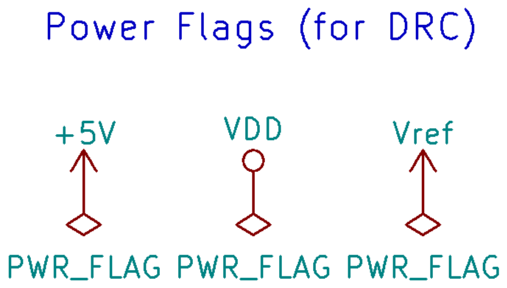
If starting from scratch you would need to draw a schematic.

For the PCB coursework the template includes the schematic. (not this one)

PCB Design Process

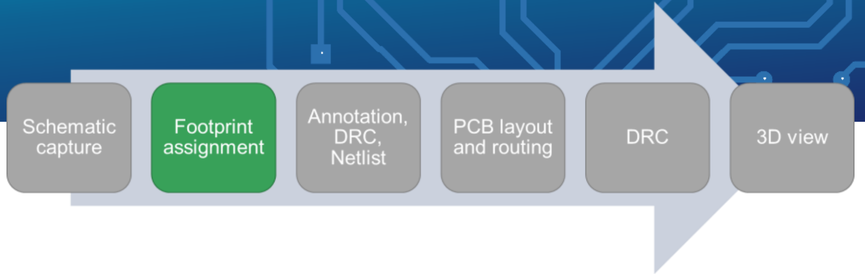


- **Through schematic capture you can:**
  - Place component symbols
  - Connect with wires – you can also use labels
  - Add 'PWR_FLAG' symbols – these indicate external power sources

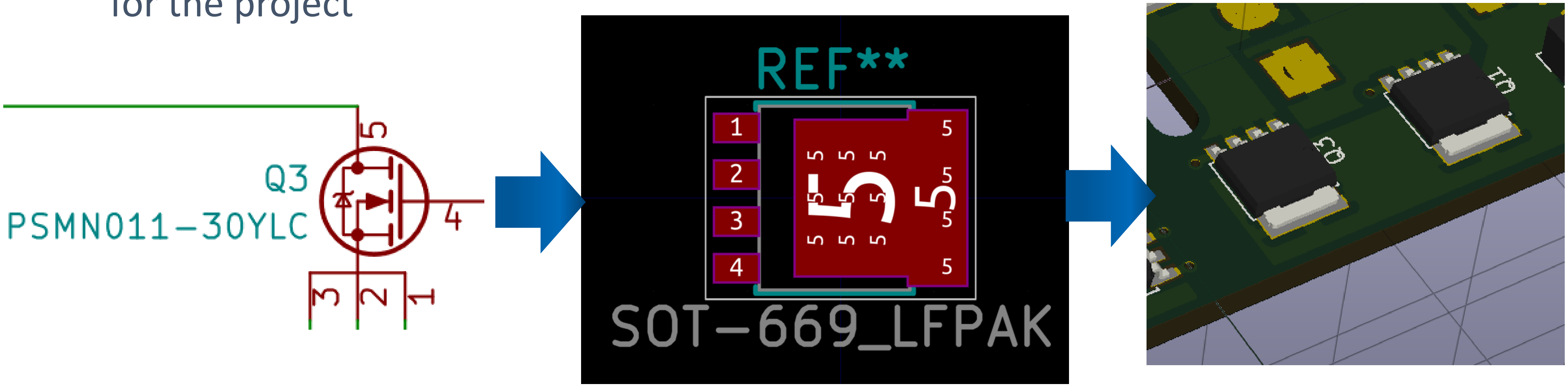


- Make sure to keep the layout neat – you'll rely on the schematic when debugging

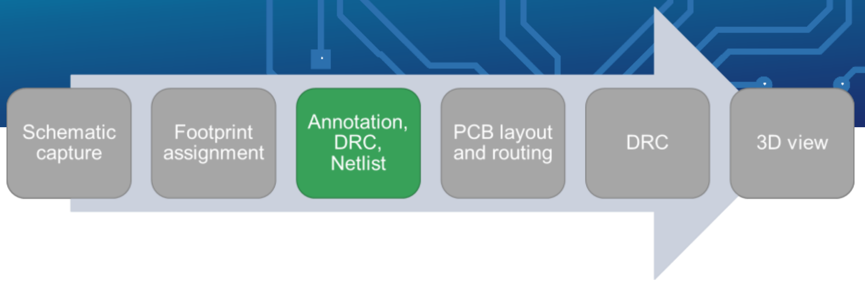
PCB Design Process



- **Component/Footprint selection:**
  - Need to link the schematic symbol to a physical layout of pads (called a footprint)
  - Must match the component you plan to use – you can confirm this by taking measurements from the part, or checking the datasheet
  - We’ve produced a custom library as part of the template that includes footprints for the project



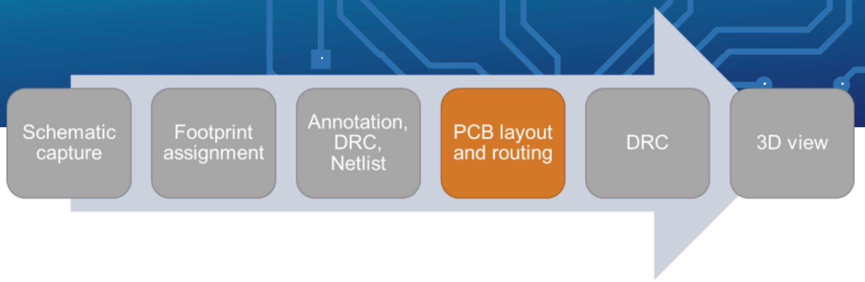
PCB Design Process



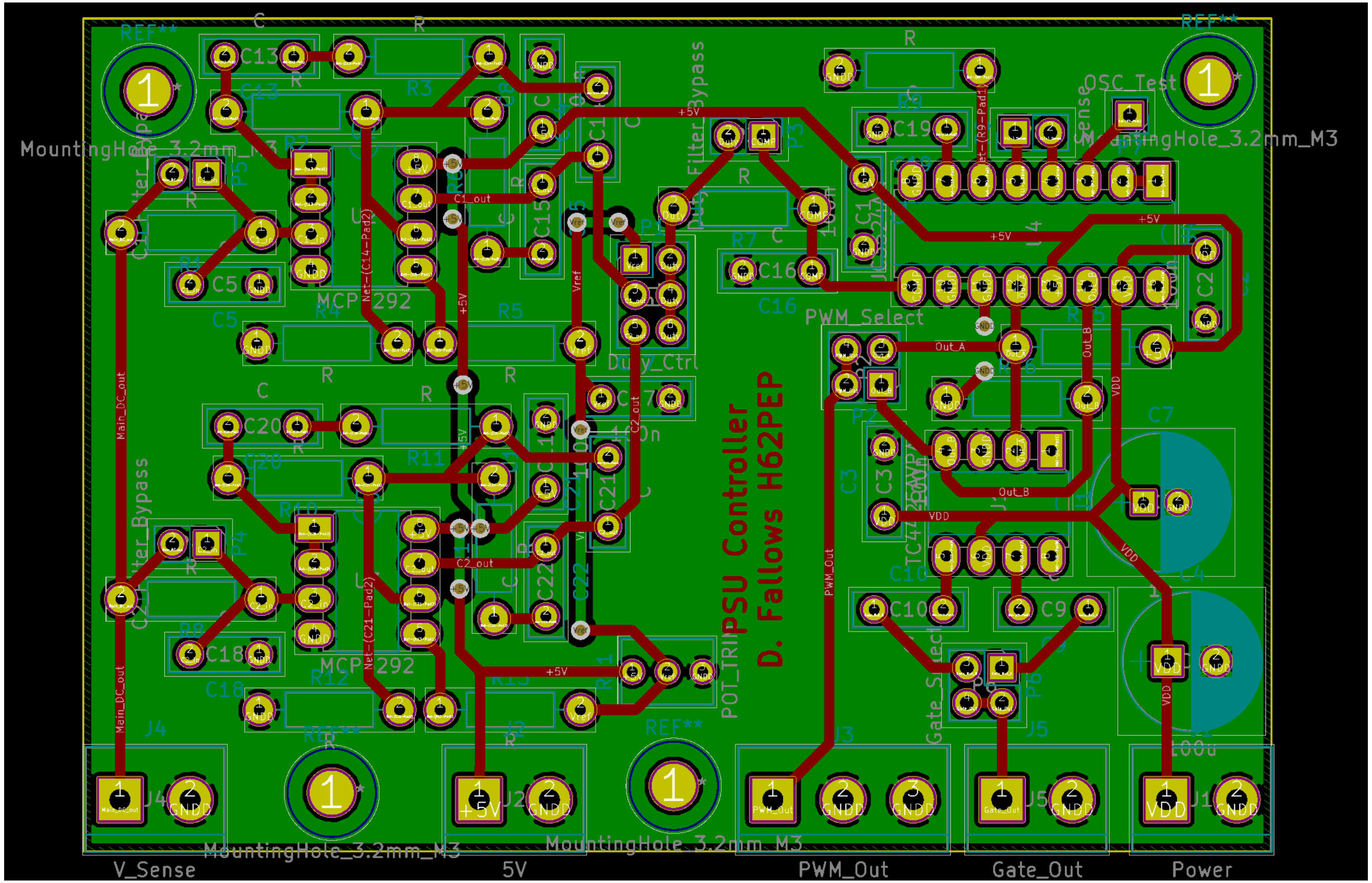
- **Annotation and DRC:**
  - Annotation assigns each component a unique letter/number combination
  - Design Rule Check (DRC)
    - Checks the schematic for simple errors (e.g. unconnected pins)
    - Does not check functionality – passing DRC **does not** mean your circuit will work

  - [In old versions of KiCAD you have to manually generate the netlist. Since version 6 this is now automatic].

PCB Design Process



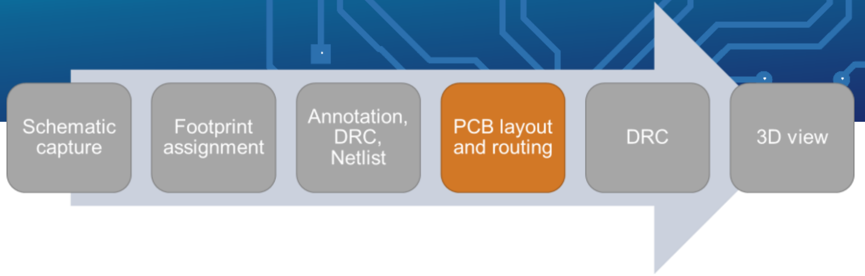
- **PCB Layout:**



This is what you need to produce – but for the main converter schematic.

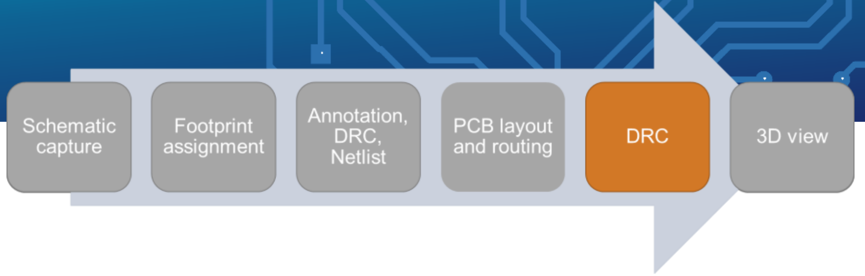
(This is the control board you'll be given in a later project week).

PCB Design Process



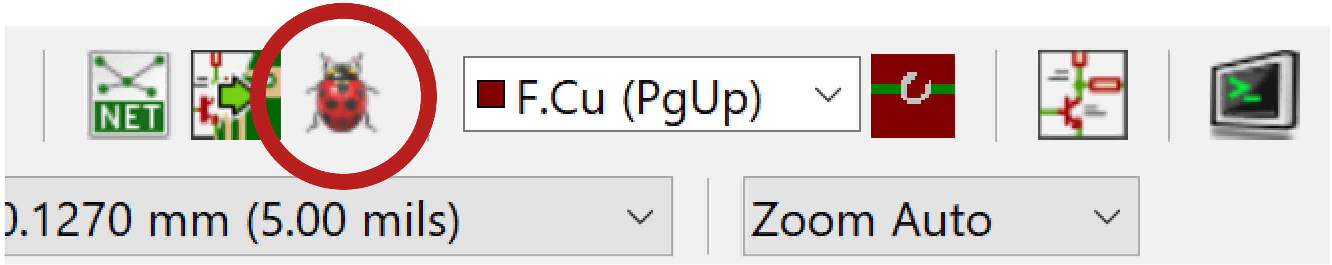
- **Basic PCB layout tips:**
  - Think about component placement
    - Will it be easy to solder? Are connectors accessible? Can test leads be easily fitted?
    - Are connected signals close by – minimise track lengths
  - Use both layers
    - For tracks and components
    - Use fill planes where possible (e.g. GND, V+)
  - Label components, connectors, signals
    - Makes it easier to construct and debug
    - Can also add test points for key signals

PCB Design Process

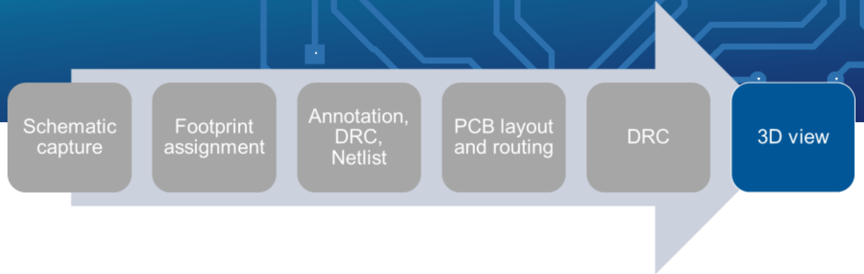


- **PCB DRC:**
  - Performs key checks
    - Are tracks too close
    - Are all the connections made
    - Are components overlapping

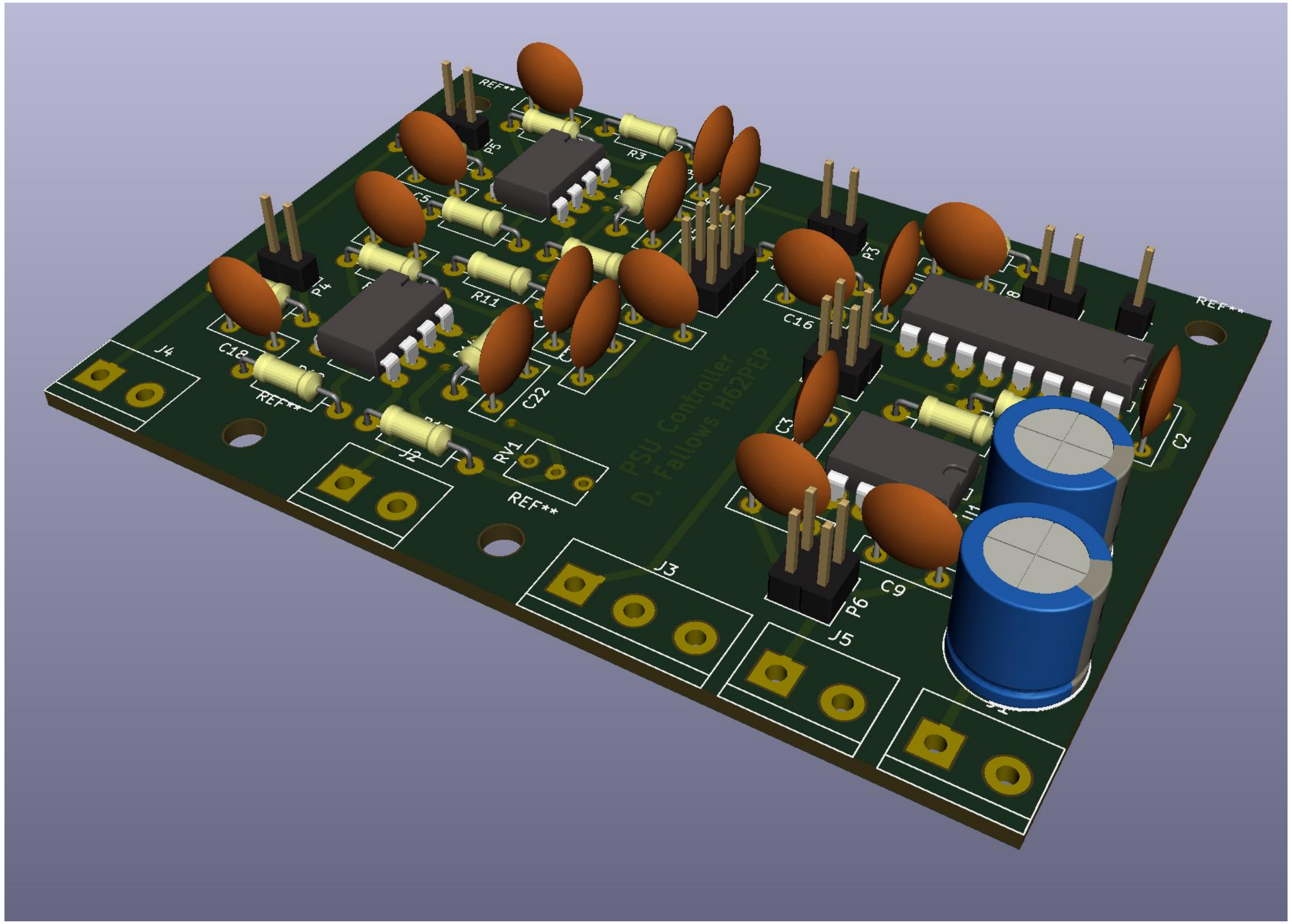
  - Does **not** confirm the design is good or will work



PCB Design Process



- **3D Rendering:**



The 3D viewer (Alt+3) will give a rendering of your design.

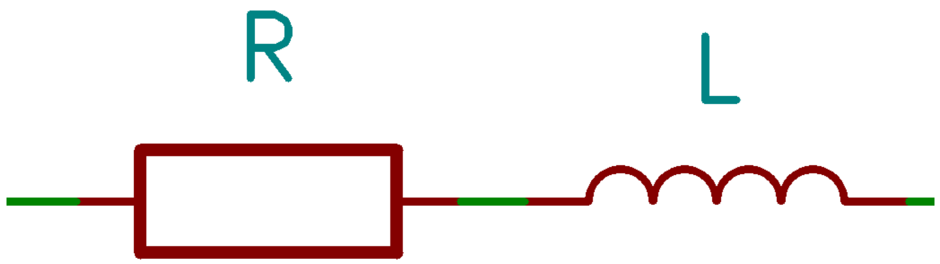
This can be helpful to check if it can be built and used easily. (Also useful to see any silkscreen mistakes)

University of Nottingham
UK | CHINA | MALAYSIA

Power Electronic Considerations

Key Considerations

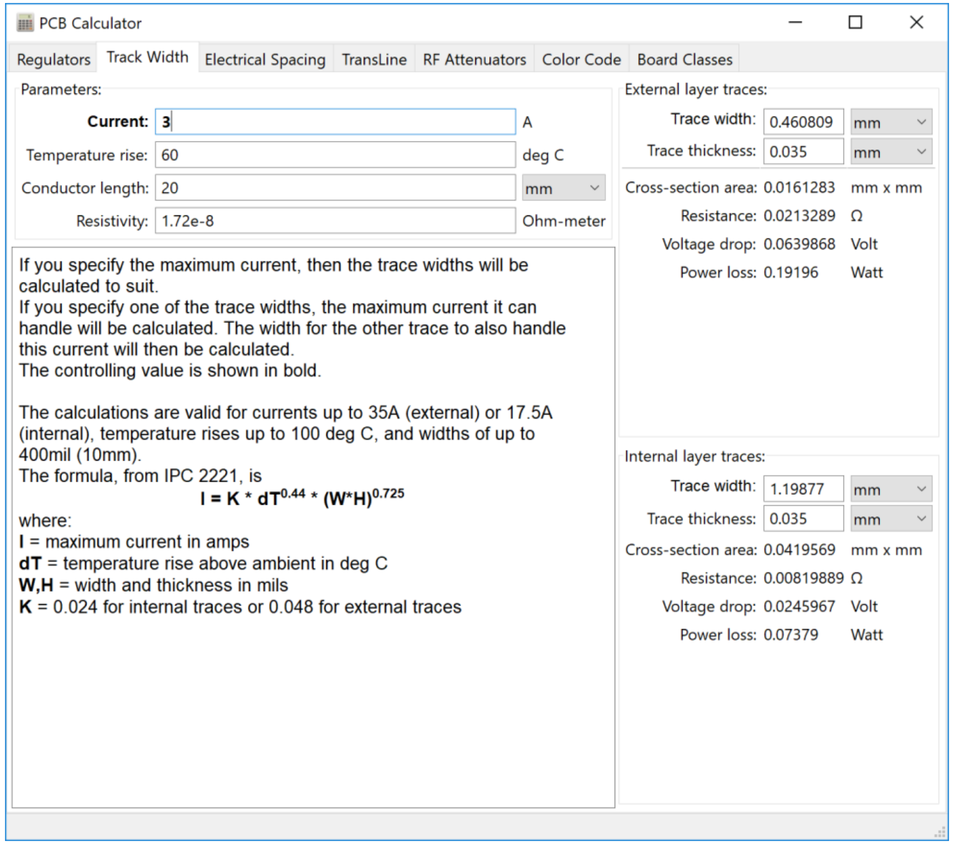
- **Power converters combine high current with high frequency:**
  - Must be considered when designing the PCB
  - Copper tracks are not perfect conductors
    - Resistance will generate heat
    - Inductance will filter high frequencies



University of Nottingham
UK | CHINA | MALAYSIA

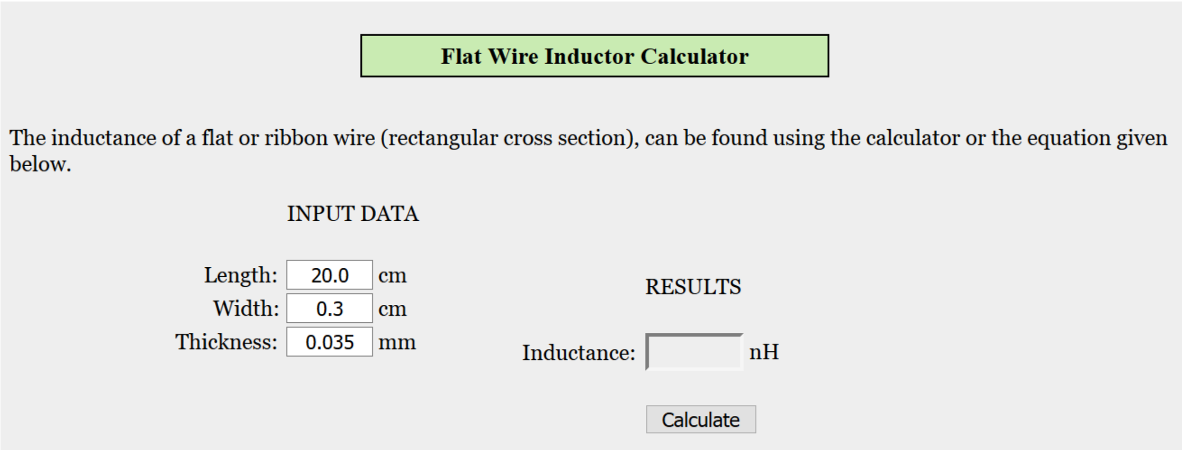
Key Considerations

- **Resistance** is controlled by track width
  - Wider track = more copper area = lower resistance
    - (just like the wire calculation for your wound components)
  - KiCAD calculator for minimum track width
    - Need to know the current and choose a suitable temperature rise
  - If using vias these must also be a suitable size
    - (and/or use multiple in parallel)



Key Considerations

- **Inductance** is controlled by track length and width
  - Short track = lower inductance, Wide track = lower inductance
  - Think about effect of additional inductance on each circuit connection
    - Is there already significant inductance present?
    - Is the connection carrying high frequency current?



Flat Wire Inductor Calculator

The inductance of a flat or ribbon wire (rectangular cross section), can be found using the calculator or the equation given below.

INPUT DATA

Length: 20.0 cm
Width: 0.3 cm
Thickness: 0.035 mm

RESULTS

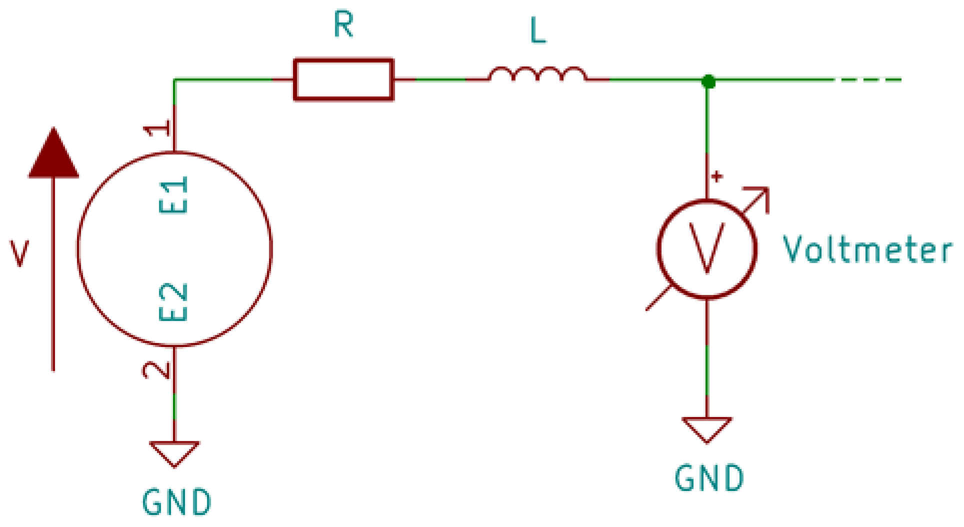
Inductance: nH

Calculate

https://chemandy.com/calculators/flat-wire-inductor-calculator.htm

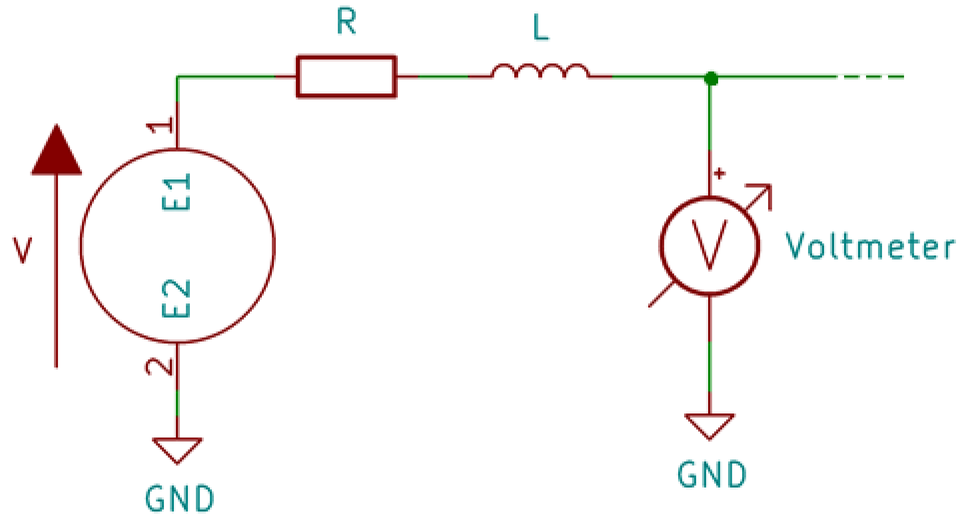
Key Considerations

- **Decoupling** (and the gate driver)
  - Power is supplied though a track
    - Voltage drop across R due to current flow (ohms law)
    - Voltage ripple across L due to change in current
  - Voltmeter reading != supply voltage
  - Need to consider the connected load
    - Does it draw a constant current?
    - If not, voltage ripple may be significant and cause malfunction
    - May exceed the supply voltage → component damage
    - May drop low causing the device to turn off or reset



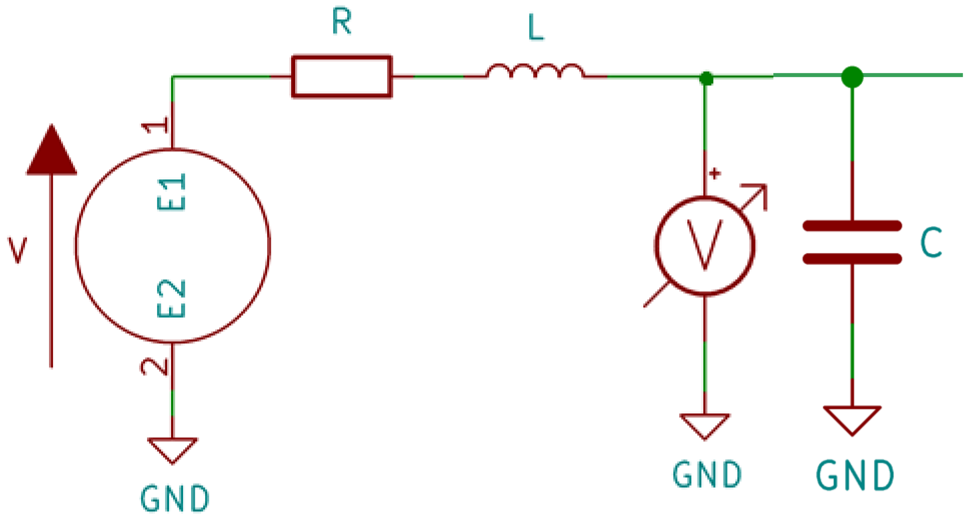
Key Considerations

- **Decoupling** (and the gate driver)
  - Solution?



Key Considerations

- **Decoupling** (and the gate driver)
  - Solution?
  - Smooth ripple by placing capacitor close to the load
  - This provides a low inductance source for high frequency current



- Why is the gate driver sensitive?
  - Draws large, high frequency switching current
    - ~100kHz
    - Rated for up to 3A (datasheet)
  - Inductance will cause large ripple
  - Decoupling must be present

Voltage ripple causes the gate driver to switch on/off quickly, switching loss then heats up the IC.

Key Considerations

- **Heatsink placement**
  - Although switch-mode power supplies are very efficient, their high power capabilities means produced heat must be managed
    - You'll carry out calculations for this in the lab
    - And measure with the thermal camera
  - Heat sinks increase surface area allowing heat to be conducted into the air.
    - Don't block the fins – air flow is required.
    - Consider the size and access
      - Must be screwed to the device
      - Is there room for a screwdriver?
      - Is there clearance for other components