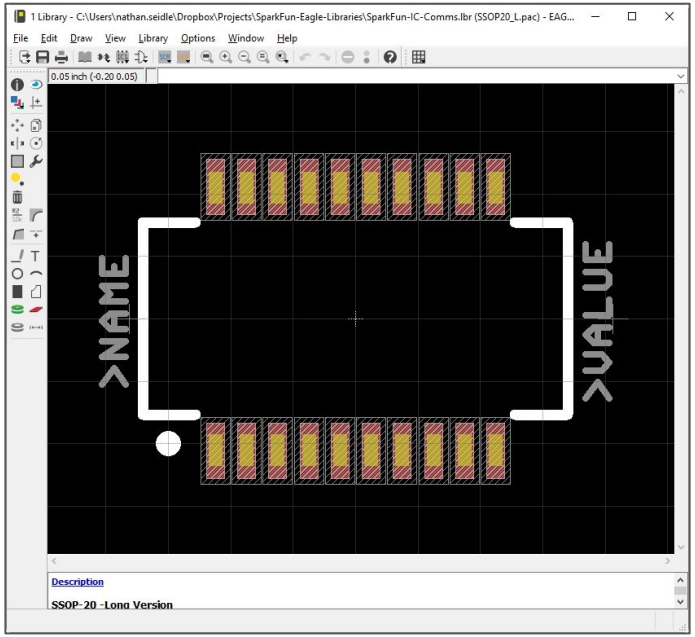
UOS EEE Society KiCAD Lectures

Component Selection and Footprint Generation

The University Of Sheffield.

Contents

1. Component Selection
    a. The difference between Theory and Reality?
    b. Considering your Application
    c. The importance of a Specification
    d. Things to look out for when choosing Components
    e. Datasheets
    f. KiCAD Component Datasheet Example
2. Footprint Generation
    a. What is a footprint?
    b. The importance of getting a footprint correct
    c. Component Dimensions
    d. Entering footprints into KiCAD Example



The difference between Theory and Reality?

Theory:

- Theoretical calculations give us exact values for components.
- This is brilliant for simulations and initial design.

Reality:

- In reality, you cannot buy components to specific values. It would be too difficult to manufacture every value anyone would ever want.
- Modern manufacturing methods also cannot provide exact component values at low cost, there is always a tolerance.

Considering your Application

- Choosing components is all about considering the application you are designing for.
- If you are designing for a low cost, low performance application; component cost is key.
- If you are designing for a high cost, high performance application; component tolerance is key.

Component selection is always a trade off. This is why it is difficult to produce a design that is low cost and high performance.

The importance of a Specification

- A good specification will inform you about the key aims of the design.
- It should be clear on the constraints you are working within, as well as the areas where you have space to make changes and expand your design.
- **It should always be used as a reference when selecting components!**

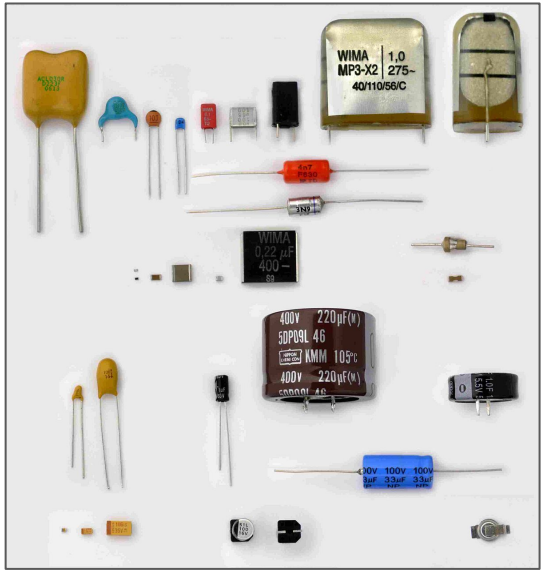
Sample Specification:

1. Switch Mode Power Supply - 20 to 30V Input - 5V Output - 30W Continuous
2. 5% Output Ripple - 2% Output Regulation (10% to 100% of Load)
3. Design should fit within 80mm x 40mm Area
4. Aiming for High Efficiency

What should we look out for when selecting a Component?

Things to look out for

- Cost
- Size and Dimensions
- Value Tolerance
- Peak Power Dissipation
- Continuous Power Dissipation
- Peak Voltage
- Peak Current
- Continuous Current
- Technology (e.g Electrolytic/Ceramic Capacitor)
- Lifetime
- Weight



Datasheets

- Datasheets are your key source of information for choosing components.
- Datasheets give a list of component characteristics, dimensions, etc.
- A good datasheet will give you all the information you need to design your circuit around the component.
- A brilliant datasheet may even give a guide of how to design a commonly used circuit around the component.

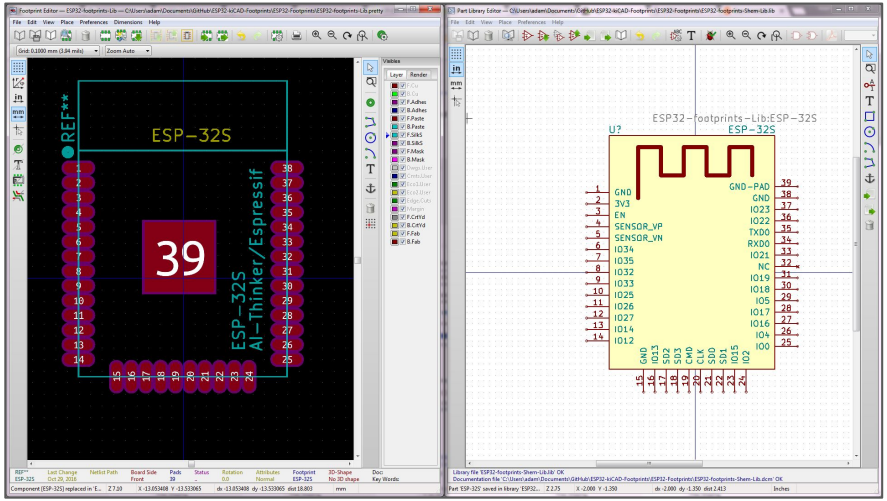
Texas Instruments provide some particularly good datasheets!

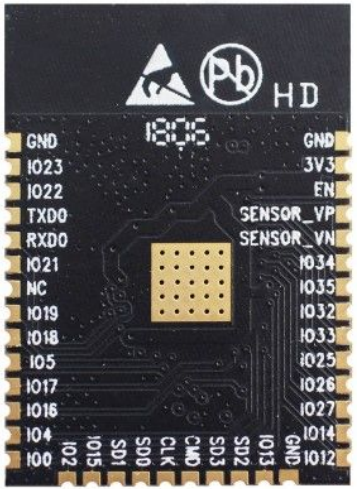
Component Datasheet Example

What is a Footprint?

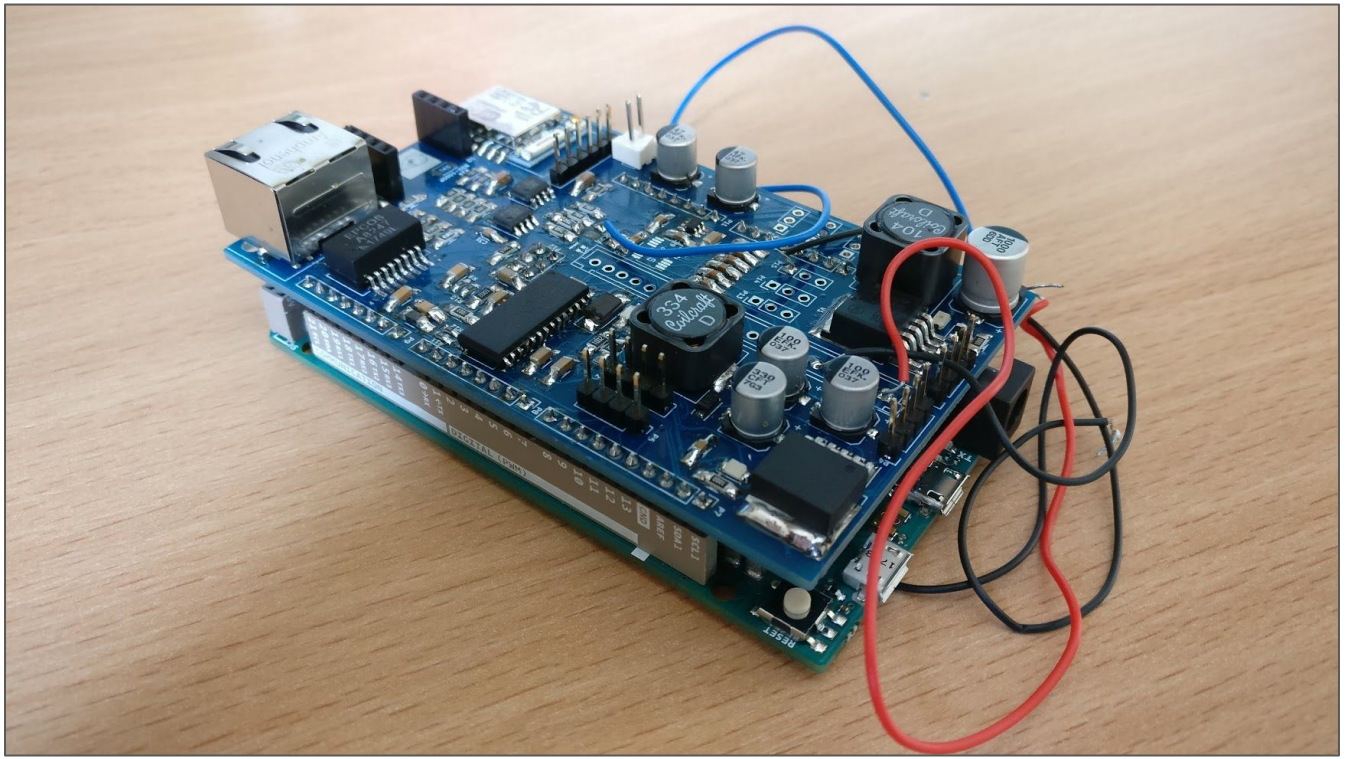
- A footprint is the physical dimension of the metal pads that are used to connect the component to the circuit board.
- Footprints come in many different shapes and sizes, depending upon the type of component and it’s specific characteristics.
- For example, 1206 is a common surface mount resistor footprint. D2PAK is a common surface mount IC footprint.

What is a Footprint?

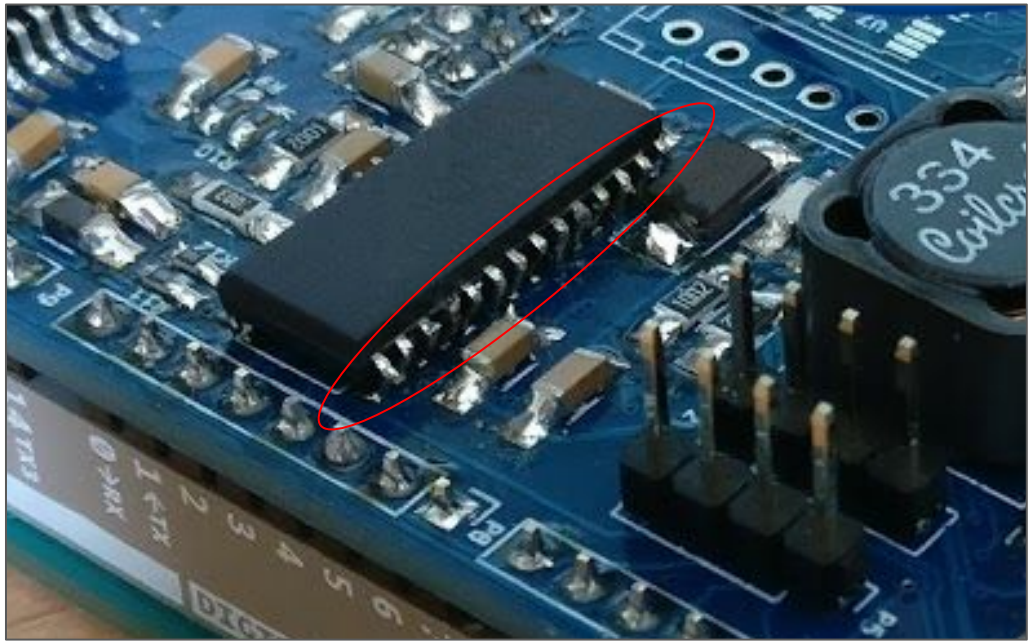




The importance of getting a footprint correct



The importance of getting a footprint correct

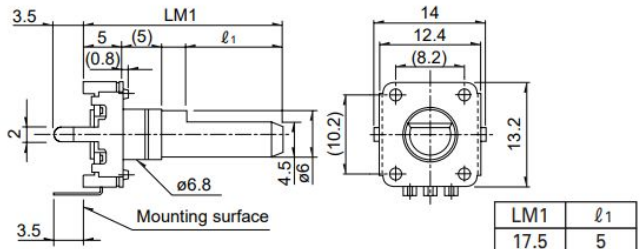
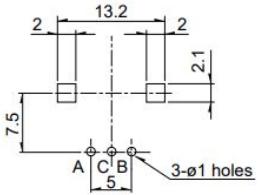


- SOIC footprint used.
- SOIC stated at the top of the datasheet.
- Footprint is incorrect!
- **SOIC Wide** stated at the bottom of the datasheet.

Check and double check!

Component Dimensions from the Datasheet

- All good datasheets will have a mechanical drawing of the dimensions of the component and its footprint.
- In rare cases you will have to purchase the component and measure its footprint yourself.

No.	Model	Style	PC board mounting hole dimensions (Viewed from mounting face)
1			

LM1	ℓ1
17.5	5
20	7
25	12

Entering footprints into KiCAD Example

Summary

- Discussed the differences between theoretical component values and real world components.
- Discussed the importance of component selection, considering the application and a good specification.
- Size, Cost, Weight, Tolerance, Power Dissipation, etc. All things to consider when selecting components.
- A footprint is the metal pads used to connect the component to the circuit board.
- Discussed the process of inserting footprints into KiCAD.
- Discussed the importance of Datasheets in Footprint Generation and Component Selection.

Thanks for listening!

Next time we will be:

1. PCB Layout Process
2. Rules of Thumb for PCBs
3. Things to consider when designing PCBs
4. Extreme Design Cases
5. Examples of PCB Layout

Follow us on Social Media:

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

Written by Sam Maxwell