

SMD PCB Workshop

Rolling Dice Game

Learn how to create Printed Circuit Boards (PCBs) the right way using KiCad software. In this workshop you will create a PCB for a rolling dice game, using Surface Mount Parts as required in higher level UQ Subjects.

Prepared by:
Tim Hadwen

1 Introduction

Printed circuit boards (PCBs) are the backbone for nearly all modern electronics. Simple PCBs are often used to help prototype a project, with more advanced PCBs used for iPhones and Televisions often requiring many more layers (in some cases over 100 layers).

1.1 PCB Concept Refresher

1.1.1 Layers

Most PCBs designed by students at University and in projects are 2 layers, top and bottom. However in some cases as size restrictions exist, more layers are required with some designs moving to 4 and 6 layers. Often if custom RF (Radio Frequency) circuitry is required, 4 layers will be required as layers on a 4 layer board are more accurate and thinner allowing the required 50Ω traces.

1.1.2 Traces

Traces are the wires that link between components on a PCB. These traces can be any width you require with standard widths being 0.2-0.3mm however high current power traces can be as thick as 1mm depending on current requirements. These traces should never have right angle bends, as this increases the resistance in the trace.

1.1.3 Planes

Planes make up large sections of copper over the PCB surface. This surface acts as an extremely low resistance trace allowing for better ground signals and in some cases high current power capacity. In 4 and 6 layer boards it is typical to have both a ground and power plane, however in 2 layer boards (like in this workshop) typically a ground plane is placed on the bottom layer. Traces should be avoided on power and ground planes to stop the breaking up of planes, although in most cases some traces will be required on the bottom layer of a 2 layer board.

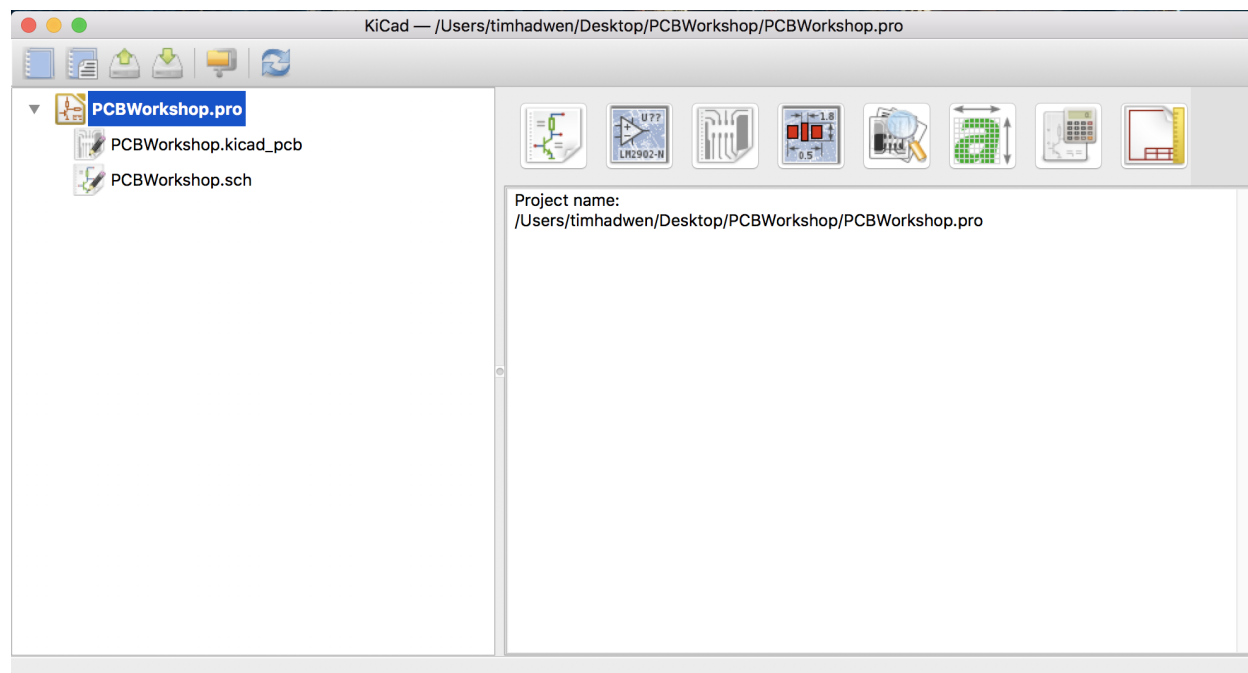
1.1.4 Footprints

Footprints are sections of the PCB that holds components, for example a footprint for a surface mount resistor is shown below, with 2 copper sections and a white line surrounding it. Often markings are added to denote what part number and what value is required in this position.

2 Workshop

2.1 Project Creation

In order to start designing a PCB in KiCad you will first need a project! To create a project click **File, New Project, New Project** and choose a name for the project.



Once the project is created we can see the project with both **kicad_pcb** and **sch** files, these are the **Circuit Board** and **Schematic** respectively. These are all the files we need to create a PCB.

On this window we can also see a series of buttons on the right. These open up the programs associated with PCB Creation. The important ones (the first 4, used in this workshop) are listed below.

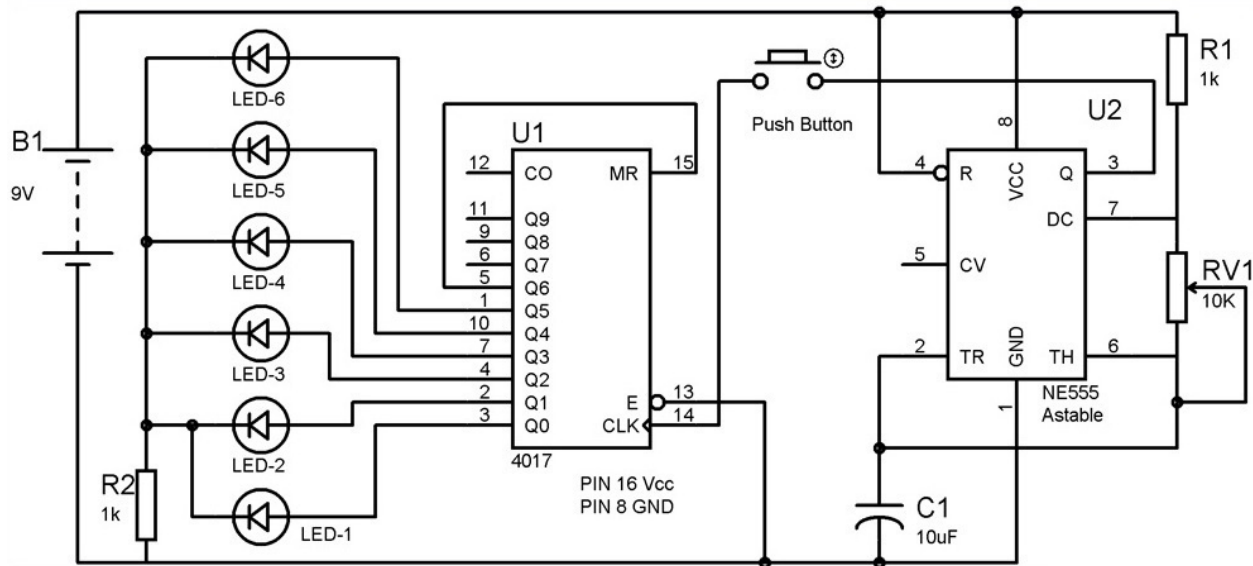
- EESchema - Schematic Creation
- Schematic Library Creator - Creates schematic symbols for use in EESchema
- PCBNew - Layout and routing of the Circuit Board once schematic is completed
- PCB footprint editor - Creates the footprints for placement on the PCB

2.2 Schematic Library Creation

In this workshop we will only be using already existing schematic parts and therefore this section will be left out of the actual workshop and is placed here for your reading later. In most cases you will be able to find the parts you need however in the off case that you can't this is the tool for you.

2.3 Schematic Creation

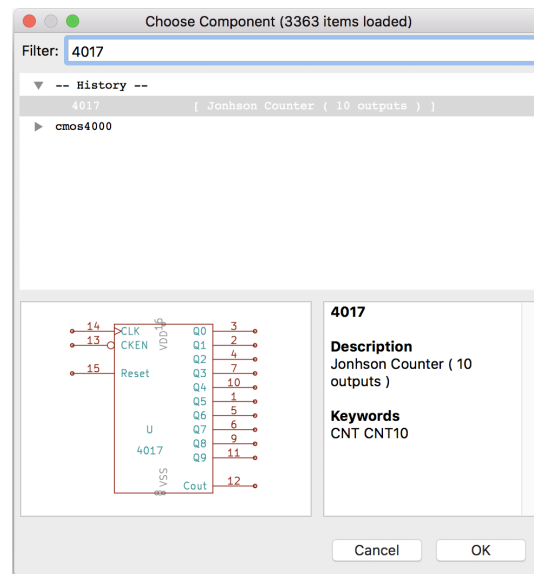
In order to create a KiCad Schematic we must first have some schematic designed. In this case we have provided you with the one below! The circuit we are building is a Electronic Dice, that on a button press will generate a random number between 1 and 6 and show that number on a series of LEDs. It does this by using a 555 timer and a CMOS Decade Counter Chip. While the button is pressed the lights will cycle until the button is released, showing a the result. The speed can be adjusted by changing the resistance of RV1, a 10k Ω potentiometer.



First, you'll need to know a few keyboard shortcuts. They are listed below. Your mouse must be hovered over the part you want to Rotate, Delete or Move.

- A - Add a schematic part
- Backspace - Delete a schematic part
- R - Rotate a part
- M - Move a part
- W - Start a wire
- K - End a wire

Now that you know that, we will add the CMOS Chip and 555 Timer. To do this press **A** on the keyboard. You can type in **4017** to get the CMOS chip to show up as a Johnson Counter as shown below.



Then, click ok and add the part to the schematic by clicking anywhere on the sheet, probably somewhere in the centre is good for this part since we will need to connect things to both sides. Remember we can always move it later!

