PRINTED CIRCUIT BOARD

(PCB)

LED Changer Circuit convert PCB using PCBWizard
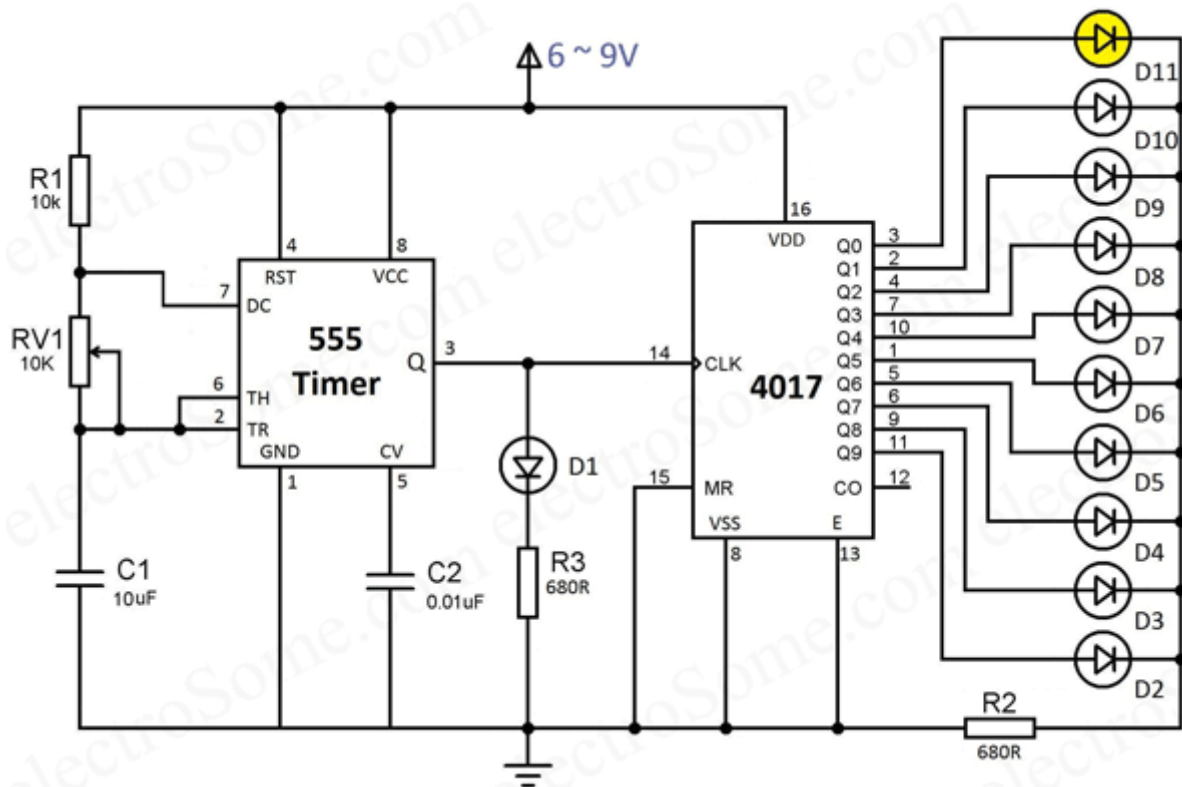
This tutorial shows you how to design and make a test probe with PCB Wizard. You should follow this tutorial to learn the basic skills you will need to use PCB Wizard effectively.

You can see that the 555 timer wired as an astable multivibrator and its output is connected to the clock input of 4017 counter IC. The output frequency of the 555 timer is determined by the resistors R1, RV1 and capacitor C1. VCC (8th pin) and GND (1st pin) is connected to the power supply. Reset (RST – 4th pin) is connected directly to the positive power supply to avoid accidental reset of 555 timer. Control Voltage (CV – 5th pin) is not used, so to avoid high frequency noises we are connecting a capacitor (C2 – 0.01µF) to the ground.

LED D1 is used to indicate the output of 555 IC and the resistor R3 is for current limiting.

Similarly, VDD (16th pin) and VSS (8th pin) of the CD4017 IC is connected directly to the power supply. Clock enable (13th pin) is an active low input, so it is connected to ground. Clock input (14th pin) is connected to the output of 555 timer. Each decoded output pins (Q0 ~ Q9) are connected to LED. Resistor R3 is used for current limiting. Only one resistor is required for limiting current through all LEDs since only one LED will turn on at a time.

Components:

To make this circuit you will need: 8-pin dual-in-line (DIL) socket 555 timer integrated circuit (IC), 4017 ic, resistors, capacitors and led's.

Adding components:

You will begin by creating a new (empty) document in which to draw your circuit. To create a new document, click on the New button or choose New from the File menu. Next you will learn how to use the Gallery to add components to your circuit. If the Gallery is not currently open, click on the Gallery button on the top toolbar to open it. Select the Circuit Symbols option. In the Circuit Symbol Gallery window, you will be able to see all the components that are available within PCB Wizard. Step 2 of 10: Adding components within the Gallery are grouped according to their function. At the top of the window, a drop-down list box allows you to select which group is shown. From the Power Supplies group, add a Battery component from the Gallery to your circuit. To do this: Move the mouse over the Battery symbol. Press and hold down the left mouse button. With the left mouse button still held down, move the mouse to drag the symbol onto the circuit. Finally, release the mouse button when the circuit symbol is in the required position. To make the 555-timer circuit you will also need several other components.

Adding a copper area:

When a printed circuit board is eventually manufactured, any space on the PCB layout not containing tracks, pads or labels, will be etched away from the copper-clad board. Each time this happens more of the etching solution is used up. To try and conserve this etching solution and reduce the cost of production, you can add areas of solid copper to limit the amount of etching that is required.

Using the mouse, draw a rectangular copper area covering the main body of the test probe circuit as shown above. You should leave a small margin (of about 0.1") between the copper area and the edge of the printed circuit board. Just as with the board outline, you will need to use an irregular shape to extend the copper area into the tip of the test probe. Click with the right mouse button over the copper area (again avoiding the objects that make up the circuit inside) and choose Polygon from the Shape menu. When you release the mouse button, the shape of the copper area will be changed. Just as with the board outline, you can adjust the shape of the copper area by clicking on it with the left mouse button and then moving one of the nodes that appear.

Adding text to the PCB layout:

Next you will use copper labels to add text to your printed circuit board. Copper labels will help you to identify your printed circuit board once it has been manufactured.

To add a copper:

 label choose Copper Label from the Insert menu. Next, click with the left mouse button somewhere on your circuit (you do not need to be very precise as you will be able to reposition the label later). A window will appear allowing you to type text for the label and specify the layer on which it will reside (see above). Enter a caption of 'Test Probe' and then click on the OK button. Your label will then appear on the circuit. Notice how the text in the label has been reversed. This is so that the label will appear the correct way around when the PCB is eventually made. Any copper labels placed on the solder side layer (the underside of the PCB) are automatically reversed. Position the label neatly on the board. Remember that as the label will be made of copper (since it is on the solder side layer), it must not overlap any existing pad or track in your circuit.

Viewing how the finished PCB will look:

 With your printed circuit board now created, you can see how it will look when made. On the left-hand side of the main PCB Wizard window, you will see the Style toolbar. This toolbar shows the different ways in which your circuit can be viewed. Click on the Real-World button. The display of your circuit will change to show you how your circuit would look if it were professionally manufactured. Next, click on the Artwork button. You now see the artwork (or mask) for your circuit. It is this artwork that you would use to make the printed circuit board. To see how a professionally manufactured circuit would look prior to the components being soldered in place, click on the Unpopulated button. Finally, try clicking on the Prototype button. This is how your circuit would look if made as a one-off prototype.