



**BSc (Hons) in Computer Engineering
Laboratory Practical**

**Experiment 3: Hand on Experience on PCB
Design and Simulation Software**

Name:	
Index No	
Intake	
Date	
Instructor Name and Signature:	
Comments	Grade

Objectives:

- Design electronic circuits using a PCB design and simulation software.
- Read the datasheet for a particular electronic component.
- Import libraries respective to components in a PCB design and simulation software.

Outcomes:

After completing this experiment, students would be able to:

- a) Understand essential electronic components and their usage.
- b) Explain steps for circuit designing in PCB design and simulation software.
- c) Read the circuit diagram and design it.
- d) Read the datasheet and modify the circuit as required.

Equipment Required:

Proteus 8 Professional

Ex 01: Installation and Introduction to Proteus 8 Professional

When building a circuit, breadboarding the circuit to test the outcomes and connections is considered the best way. But with this method, it is easier to get confused when it comes to larger and more complicated circuits hence making the process more tiresome and time-consuming. In situations as such, **simulating the circuit** to test the connections and components is proven to be the best and most convenient solution.

Currently, there are some simulation tools such as Eagle, Altium Designer, Proteus, and EasyEDA that can be used to simulate electronic circuits. Among them, **Proteus 8 Professional** will be the one taught in this experiment.

Proteus 8 Professional consists of several features including Among the many important uses of this software, Schematic Drawing, Simulation, PCB Designing, 3D Visualization, and Billing. This practice will be focused on Schematic Drawing and Simulations.

Layout for the Schematic Layout window in Proteus.

Figure 1 shows the layout for the Schematic Layout window in proteus which we will be used to observe the simulations.

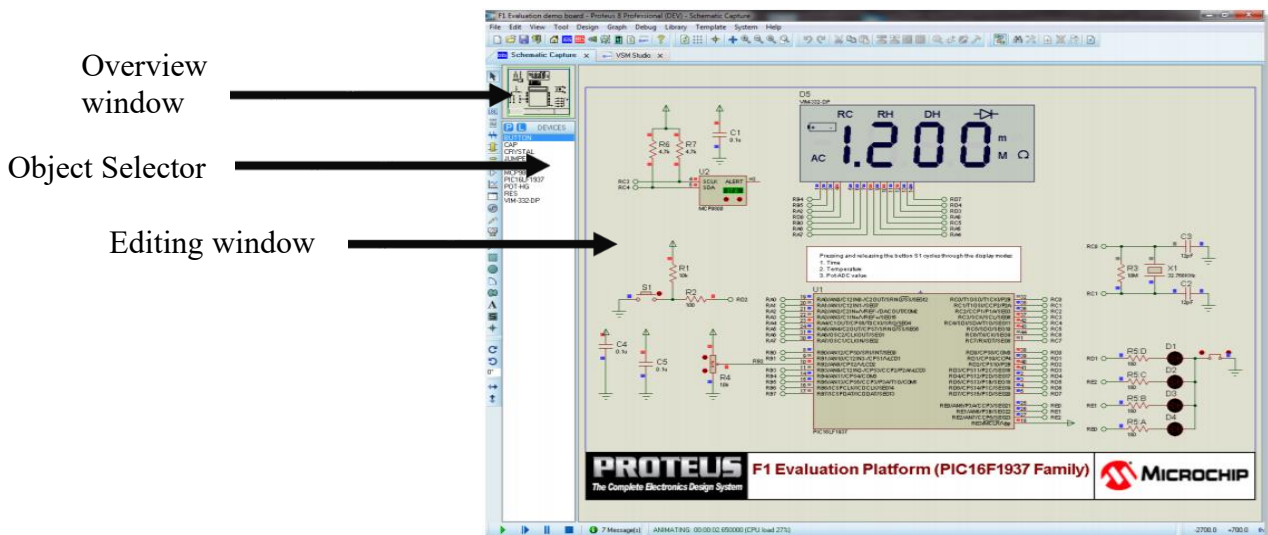


Figure 1: Proteus Window

The 'P' and 'L' buttons give us to find and place components and import device libraries, respectively. Once the circuit is built and wired, pressing the play button in the left bottom corner of the window will allow us to see the simulation of the out circuit. (Figure 2)



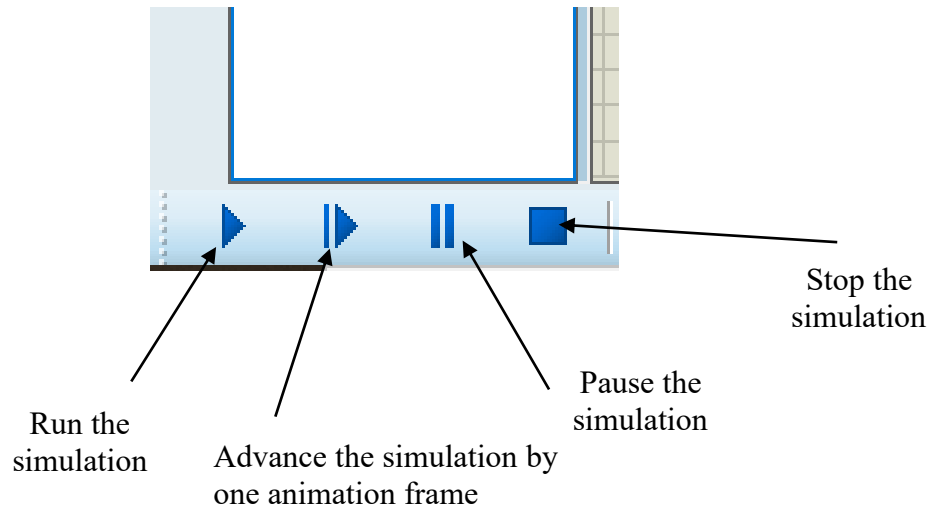


Figure 2

Ex 01: Design and simulate the Multivibrator Circuit

Before creating your projects, this exercise uses to learn the layout, functioning, component placement and the outlook of the proteus software.

Steps

1. Create a “New Project” with a schematic layout and save it to a location convenient to you.
2. Find select and place components in the schematic window by observing the diagram given.
3. Make the wiring connections to complete the circuit.
4. Place an oscillator with terminals at the two ends of the capacitor, to observe the waveforms respective to the functioning of the capacitors.
5. Convert the Schematic diagram to its PCB layout. You may use auto-routing as well as manual routing to understand the PCB connections.
6. Generate and observe the 3D diagram for the PCB for better understanding.

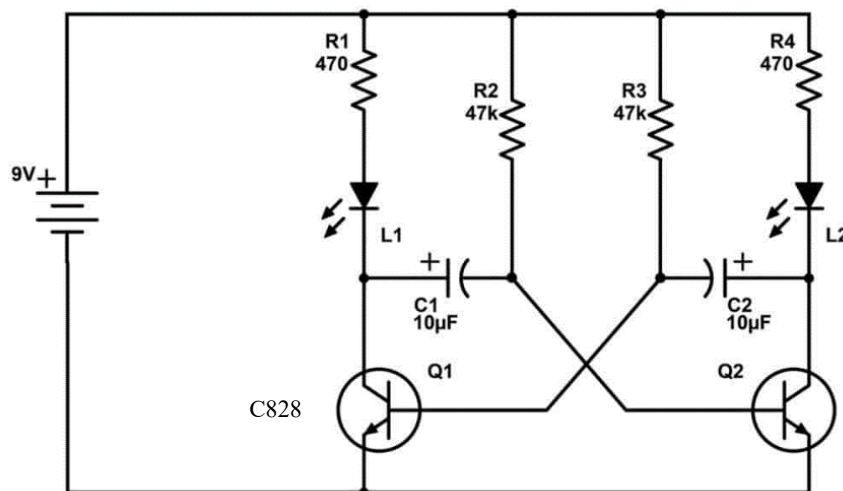


Figure 3. The Circuit of the Multivibrator

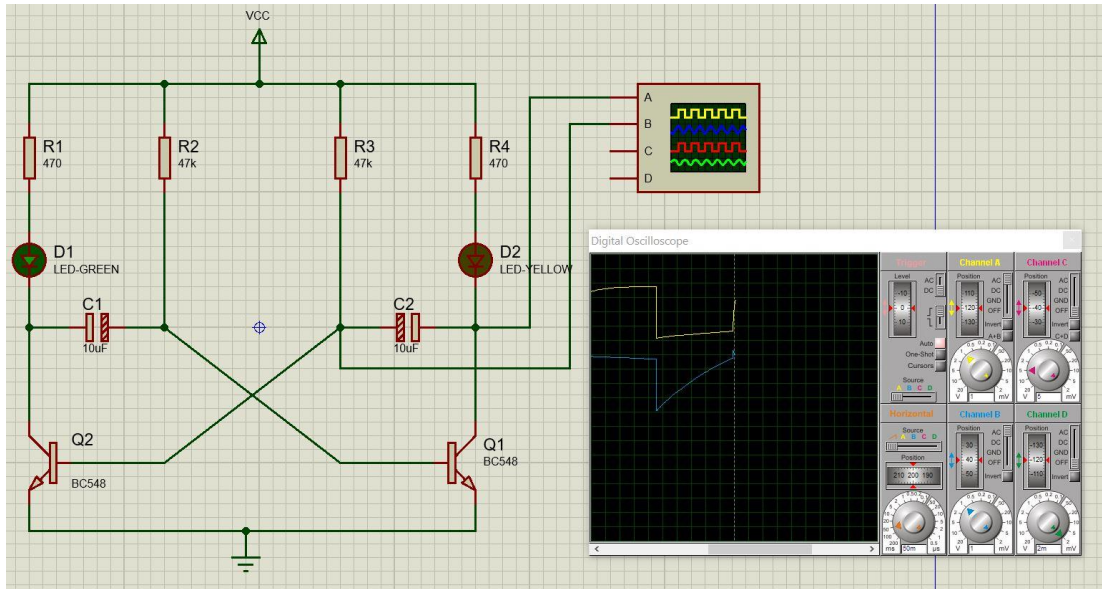


Figure 4: Schematic View of the Circuit with Simulation

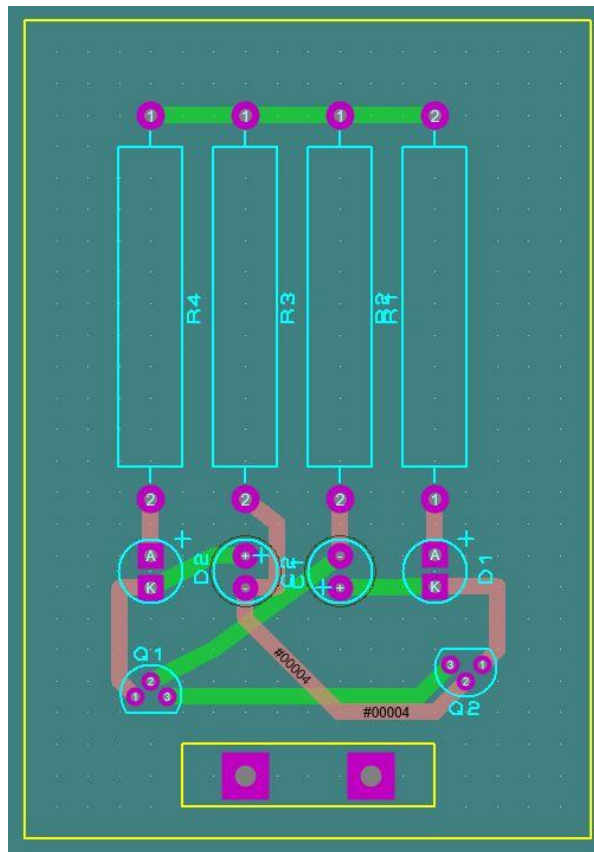


Figure 5: PCB Layout of the Multivibrator Circuit

Read the datasheet and implement circuits as required

There are dedicated IC's specially designed to accurately produce the required output waveform with the addition of just a few extra timing components.

One such device that has been around since the early days of IC's and has itself become something of an industry "standard" is the 555 Timer Oscillator which is more commonly called the "555 Timer".

Monostable 555 Timer

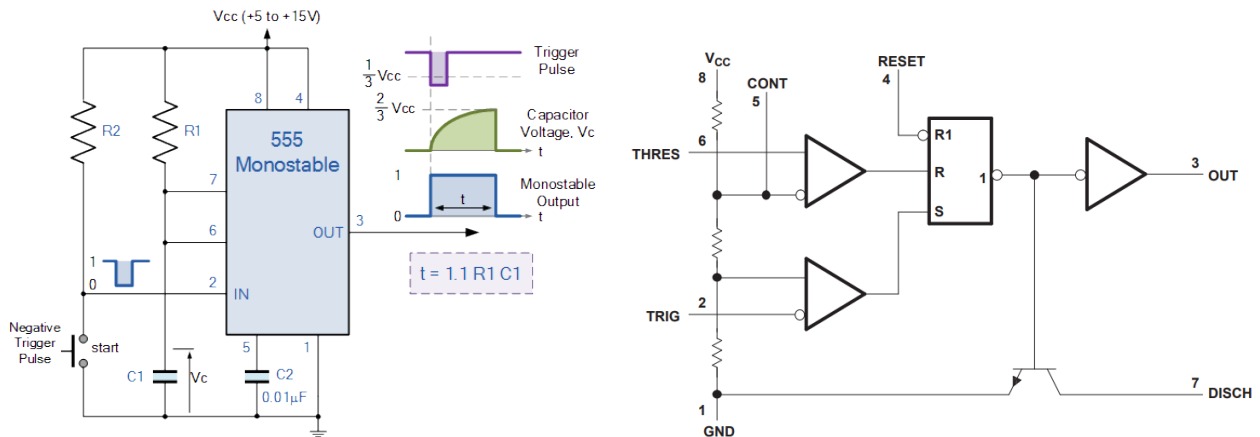


Figure 7: 555 Timer

Ex01: Read the datasheet clearly and implement a timer using NE 555

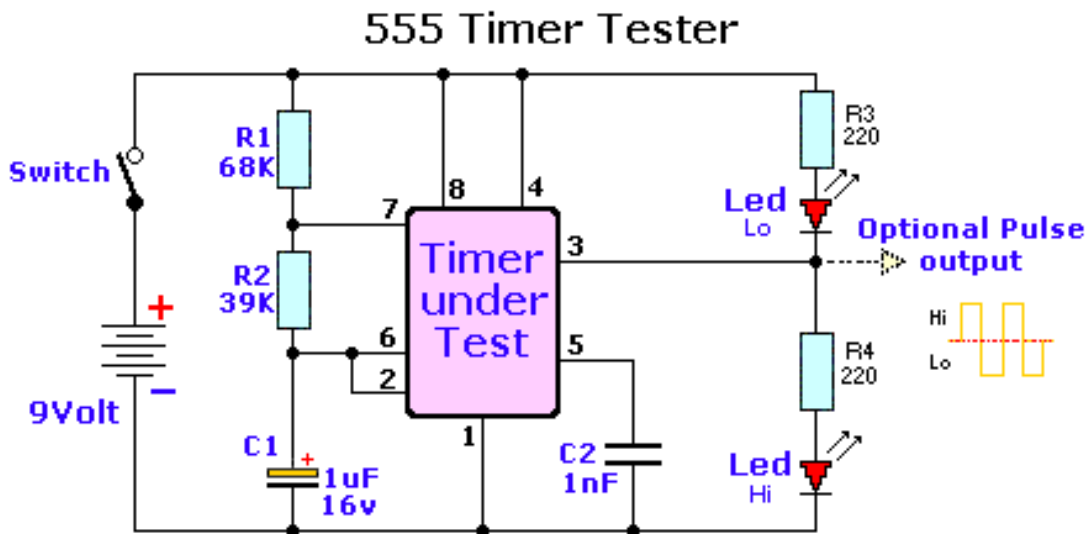


Figure 8: Multivibrator using 555 Timer

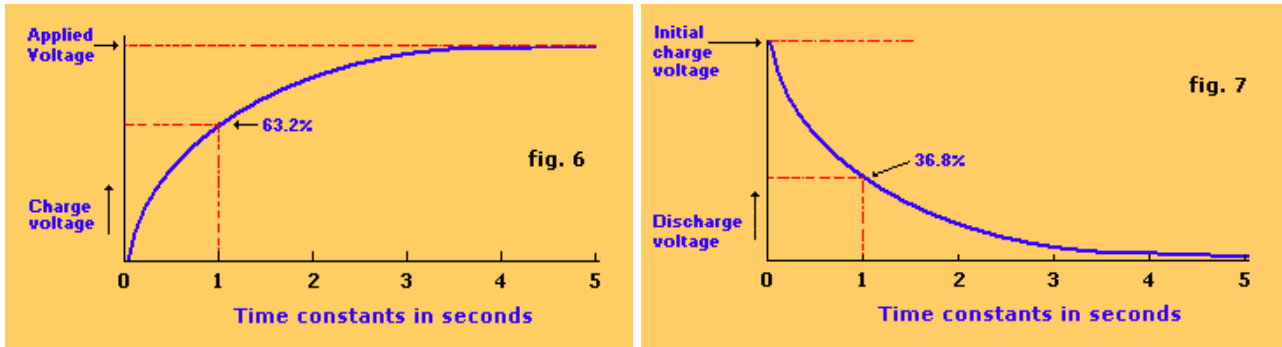


Figure 9: Graph for charging and discharging

Note:

The frequency of operation of the circuit is dependent upon the values of R1, R2, and C. The frequency can be calculated with the formula:

$$f = 1/ (.693 \times C \times (R1 + 2 \times R2))$$

Calculate suitable RC values to create 1Hz 10 Hz and 100 Hz oscillation (Use existing R or C value first and calculate suitable other value)

.....

.....

.....

.....

.....

.....

.....

Timer Application

The CD4017 is a CMOS Decade counter IC. CD4017 is used for low-range counting applications. It can count from 0 to 10 (the decade count). The circuit designed by using this ic will save board space and also the time required to design the circuit. CD4017 is as ‘Johnson 10 stage decade counter’.

1. Implement the following “Nightrider circuit” using 555 and 4017

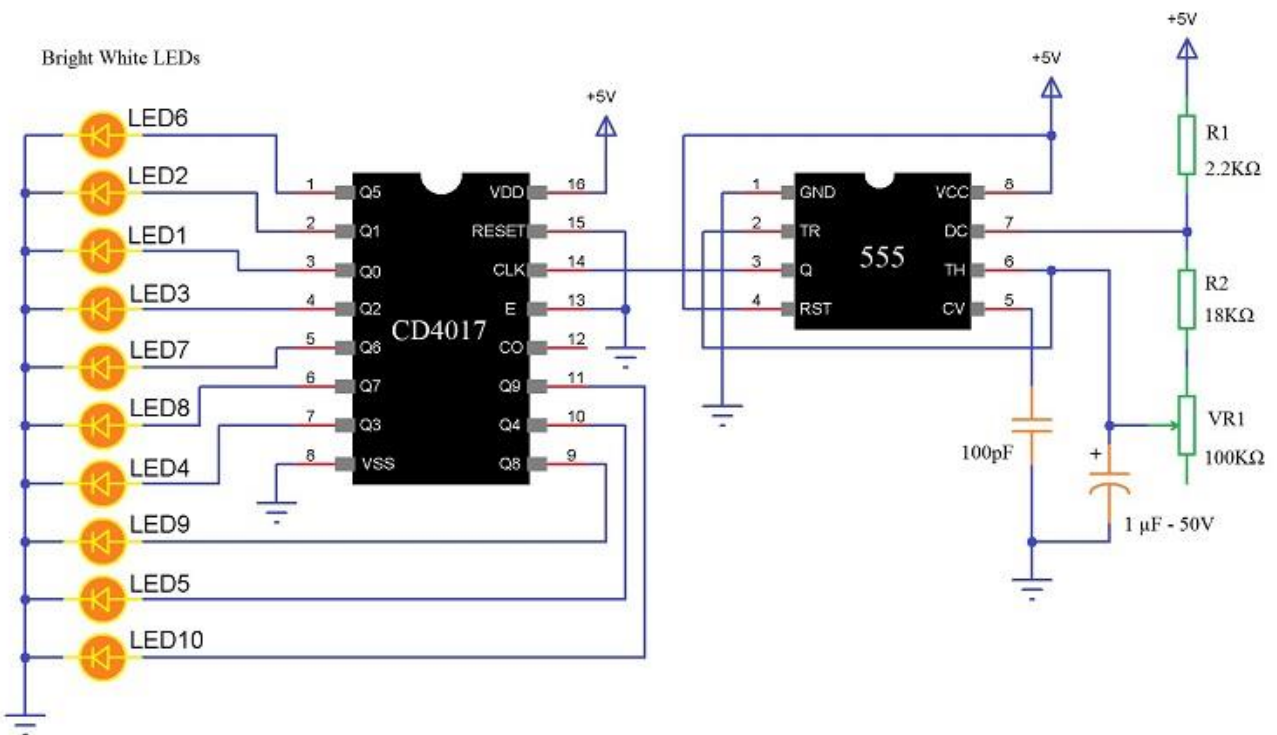
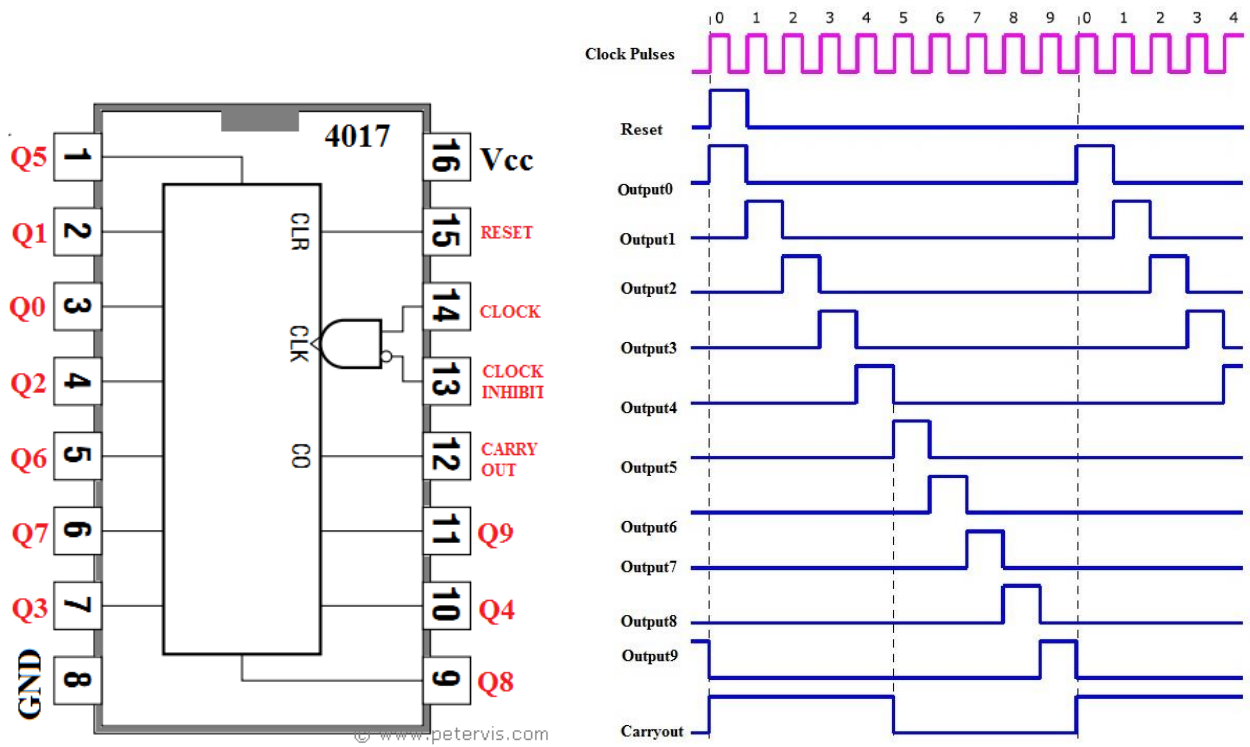


Figure 10: Knight rider Circuit

Modify your circuit as the flash LEDs on Both directions (“Like Knight Rider”)

Then explain a way to modify the above circuit to flash 20 LEDs using two 4017 ICS

END