

## **Experiment 2: Lab Manual**

For the given the GitHub Code for Experiment 2 (To perform nodal analysis for a given linear dc circuit using SPICE)

- 1) Reconstruct/Draw the circuit.
- 2) Simulate the Circuit. Report all the node Voltages.  
V(1)=\_\_\_\_\_  
V(2)=\_\_\_\_\_  
V(3)=\_\_\_\_\_  
V(4)=\_\_\_\_\_
- 3) Verify it: Solve it Manually

### **Experiment 7: Lab Manual**

For the given the GitHub Code for Experiment 7 (To perform nodal analysis for a given linear dc circuit using SPICE)

- 1) Reconstruct/Draw the circuit.
- 2) Simulate the Circuit. Report all the branch currents.  
     $I(v3)=$  \_\_\_\_\_  
     $I(v4)=$  \_\_\_\_\_
- 3) Simulate the Circuit with  $v1=0v$ ,  $v2=10v$  and report the following branch currents.  
     $I(v3)=$  \_\_\_\_\_  
     $I(v4)=$  \_\_\_\_\_
- 4) Simulate the Circuit with  $v1=10v$ ,  $v2=0v$  and report the following branch currents.  
     $I(v3)=$  \_\_\_\_\_  
     $I(v4)=$  \_\_\_\_\_