

Experiment 1a: Lab Manual

- 1) Given the GitHub Code for Experiment 1a (To simulate a Linear DC Circuit for determining all node voltages using SPICE) Reconstruct/Draw the circuit.
- 2) Simulate the Circuit. Report all the node Voltages.
V(1)=_____
V(2)=_____
V(3)=_____
V(4)=_____
V(5)=_____
V(6)=_____
- 3) Verify it: Solve it Manually