

Homework #1

Show all of your work. Problems are from the text book *Engineering Circuit Analysis, seventh edition*, William H. Hayt, Jack E. Kimmerly, and Steven M. Durbin. If PSpice simulation is required, show your schematics and explain simulations.

1. 3.14 (Chapter 3, problem 14)
2. 3.22
3. 4.7 Just write equations, you do not need to solve them.
4. For the previous problem, use PSpice to simulate the circuit.
5. 4.9. Just write equations, you do not need to solve them.
6. For the previous problem, use PSpice to simulate the circuit.
7. 4.31 Write equations only.
8. 4.37.