

EECE 311 Laboratory Project #2

Instructor: Dr. Hede Ma

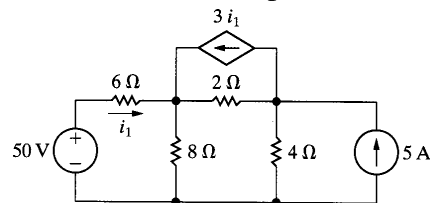
Use SPICE for Nodal Analysis

1. Objective

The objectives of this laboratory exercise are to learn and gain experience in analyzing and simulating electric circuits using PSPICE.

2. Laboratory Procedure

1. Generate PSPICE input file for the circuit to find its nodal voltages.



2. Save your file with your initials.
3. Run PspiceA_D and open the file.
4. After Simulation Completed Successfully, go File → Examine Output → Print.