

# EECE 311 Laboratory Project #3

Instructor: Dr. Hede Ma

## Use SPICE for Electric Circuit Analysis

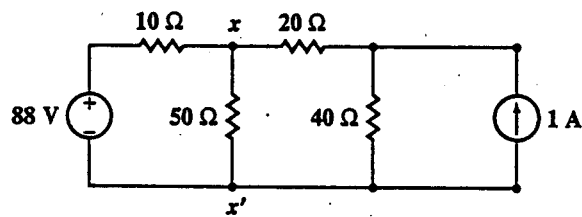
### Thevenin Equivalent Circuit

#### 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 shown as follows to find the Thevenin equivalent circuit from the terminals  $x$  and  $x'$ .



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.