Experiment 5
Schematic Capture with LTspice

Introduction:
This experiment is based on an interactive study of Appendix M.2 of the EE 2240L Laboratory Manual. That document is not a comprehensive introduction or a reference manual. Instead, it focuses on some of the specific features of LTspice that are useful for electrical engineering classes. The lab session has one main goal: to assist students in becoming familiar with using LTspice with schematic capture to solve application problems. It is assumed that the students conducting this experiment have no prior experience with LTspice schematic capture.

The availability of technical computing environments such as LTspice is reshaping the role and application of computer laboratory projects so as to involve students in more intense problem-solving experiences. It also provides an opportunity to easily conduct numerical simulation experiments and to tackle realistic and more complicated problems.

This experiment is designed to be completed by students while working at a computer. The emphasis here is on "learning by doing". The experiment is intended to be fully completed in one 150-minute laboratory class.

Preliminaries:
Read and understand Appendix M.2 of the EE 2240L Laboratory Manual.

Procedure:
Complete the attached Exercises with LTspice using schematic capture for the circuit description input. Document and submit a brief summary of your results; a detailed description of your procedure is not required.
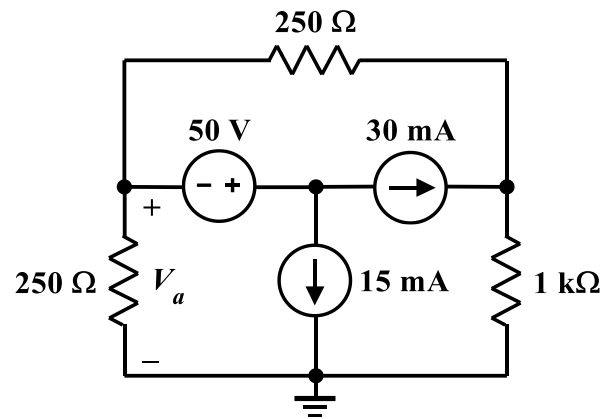
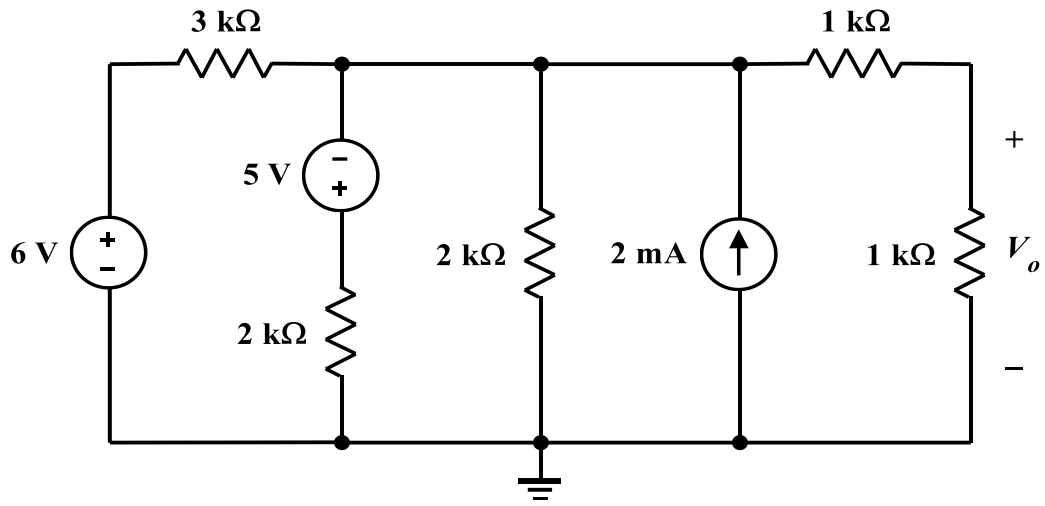**Exercise 1:** Use LTspice to determine the numerical value of V_a.

10k Ω

75 V

5k Ω

V_a

15k Ω
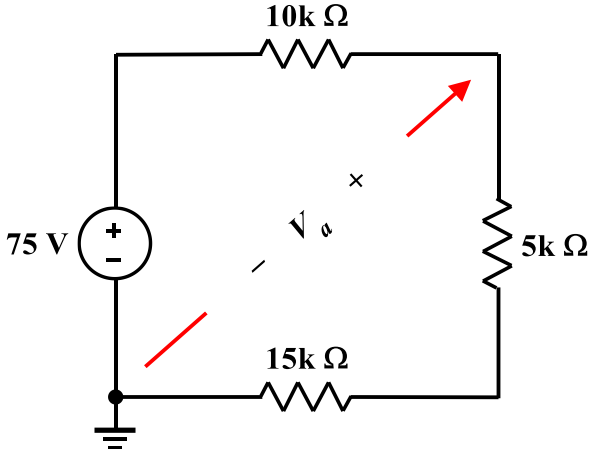
Exercise 2: Use LTspice to determine the numerical values of V_x and I_y.
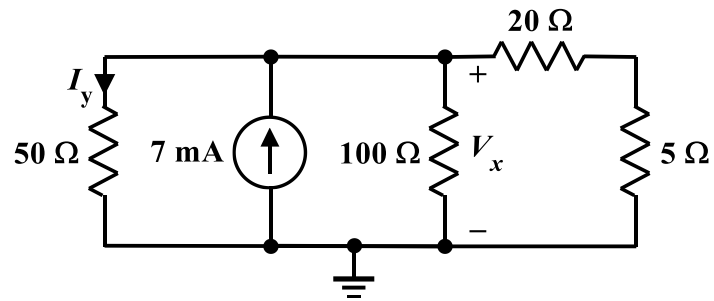
Exercise 3: Use LTspice to determine the numerical value of V_a.

Exercise 4: Use LTspice to determine the numerical value of V_o and the amount of power absorbed by the 3 kΩ resistor.

Exercise 5: Test your ingenuity – Use LTspice to determine R_{eq}.