ECE-311 (ECE, NDSU)
Lab 1 – Simulation
LTSpice – A high performance SPICE simulator

1. Objective

The objective of this lab is to learn how to use LTSpice, a high performance SPICE simulator. It is a professional grade simulator from Linear Technology. It is simple to use, free, and does not have any limitations in terms of the maximum number of components, nodes, etc. You will learn it by creating and simulating several circuits.

2. Description

As you know already from EE-206, SPICE (simulation program with integrated circuits emphasis) is a powerful computer software program used for rapid simulation and analysis of circuits. It is now an industry standard. Many EDA (electronic design automation) companies offer variations of the basic SPICE -- in most of the cases they are not free. Most of these variations are equipped with a GUI (graphical user interface). There are also SPICE variations which are free. In this course you'll use LTSpice, which is available in the computer-cluster room. Because it is free, you may want to download and install it on your personal computer if you want to use it at home. You can download LTSpice from:
http://www.linear.com/designtools/software/index.php

To learn how to use LTSpice you can read any or all of the following:

- LTspice Users Guide (downloadable from the above website)
- LTspice Getting Started Guide (downloadable from the above website)
- LTSpice tutorial from Laurier University (http://denethor.wlu.ca/ltspice/)
- LTspice courseware and tutorials from LTWiki (http://ltwiki.org/index.php5?title=SPICE_and_LTspice_Courseware_and_Tutorials)

In addition, you are encouraged to search additional resources on the Internet on your own.

You will create and simulate in LTSpice the following circuits:

- Circuit from Figure 4.31 (page 104) from textbook.
- Circuit from Figure 5.8 (page 129) from textbook.
- Circuit from Figure 6.32 (page 199) from textbook
- Circuit from Figure 8.10 (page 264) from textbook

3. Lab report

Your report should contain the following:

- At least half a page with a description of what LTSpice is. Your description should be written such that a general audience would be able to understand it (should be coherent, concise, and not use abbreviations).
- Printouts of the circuit schematic and relevant plots and waveforms for each of the circuits.