Homework Assignment #1: Boost Converter Simulation

Introduction

This assignment is an introductory simulation problem, concerning LTspice simulation of a boost dc-dc converter. To get started, do the following:

1. Download and install LTspice on your computer. Windows and Macintosh versions of LTspice IV are available at the following site:
   http://www.linear.com/designtools/software/
2. Download and unzip the boost converter and associated files from the link below, and save them in a working directory on your local disk.
   Boost.zip
3. Open the file boost.asc using LTspice. Press the run button to run a simulation of the turn-on transient of this circuit. It should take 10-20 seconds for the simulation to run (depending on your computer speed). When the simulation is done, you can click on a node in the schematic to plot the voltage at that node. If you move the mouse over an element such as the inductor, it turns into a current probe symbol; you can then click to plot the current in that element. Explore the waveforms of the converter, including the output voltage, the inductor current, and the switch node voltage.

Questions

For the numerical answers below, you should enter your answer with an accuracy of +/- 1%. For efficiency, your answer should have an accuracy of at least +/- 0.1% of efficiency.

1. What is the steady-state average output voltage (expressed in volts)?

2. What is the steady-state average inductor current (in amps)?

3. What is the steady-state output power (in watts)?

4. What is the average power drawn out of the input source Vg during steady-state operation of the converter (in watts)?

5. What is the average power consumption of the gate driver (in watts)?

6. What is the converter efficiency (enter a numeric value between 0 and 1)?

7. Now change the control voltage input to the pulse-width modulator, so that it produces a control signal having a duty cycle of 0.6. Run the simulation again. What is the new steady-state average output voltage?