

1-3 Simulation of a Buck Converter Using LTspice

1. Installing LTspice and course simulation files
2. Simulation of a buck converter
3. Homework assignment: simulation of a boost converter

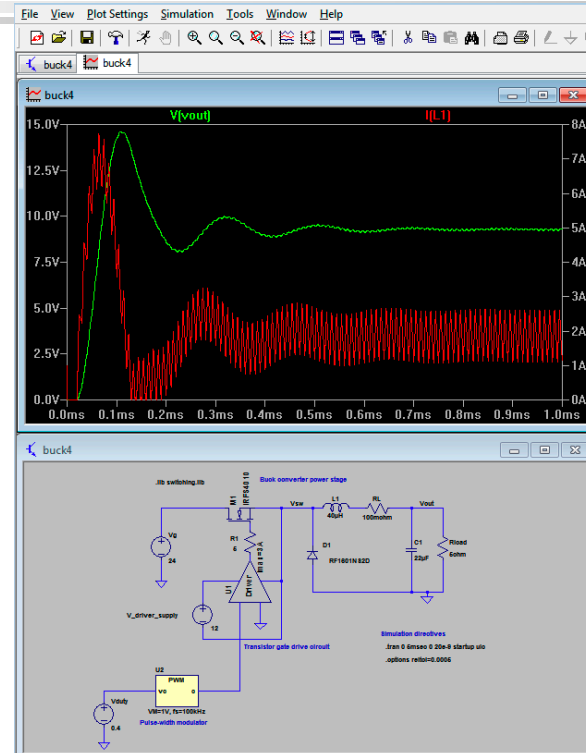
SPICE

Simulation Program with Integrated Circuit Emphasis

- Enter a circuit, then run a simulation that plots voltage or current waveforms, and computes other quantities of interest

In this course, we will use a version of SPICE that is available for free from the Linear Technology Corp., *LTspice*.

Versions of LTspice IV are available for both Windows and Macintosh systems, along with a *getting started* guide and examples.



Getting Started

1. Download and install LTspice IV—it can be downloaded from the following site:

<http://www.linear.com/designtools/software/>

2. Download the buck converter zip file from Coursera, linked to this lecture. Unzip this file and save in a folder on your disk
3. Double-click on buck.asc to open the file in LTspice. Then press the run button to start the simulation.
4. To display a waveform, click on a node in the circuit diagram to display its voltage (with respect to the circuit ground), or click on an element to display its current (with respect to the direction shown while clicking).

Pulse-Width Modulator Block

The pulse-width modulator (PWM) converts an analog input voltage v_c into a control switching waveform c whose duty cycle d is dependent on v_c .

Block functionality:

$$D = (v_c - V_{\text{offset}}) / V_M \quad (\text{duty cycle of } c(t))$$

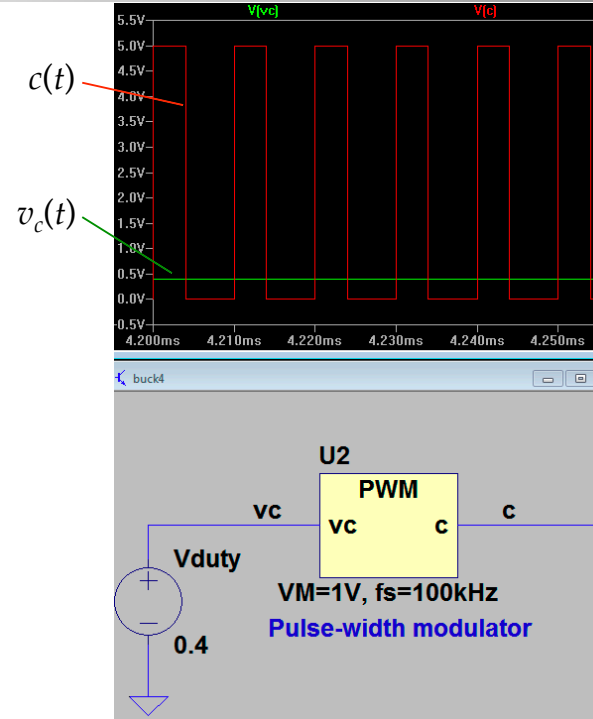
Frequency of $c(t)$ is f_s (switching frequency)

$$D_{\text{min}} \leq d \leq D_{\text{max}} \quad (\text{duty cycle limits})$$

In LTspice, the above parameters can be entered in a window obtained by right-clicking on the PWM block (or command-click on Macintosh)

In the example at right, $v_c = 0.4 \text{ V}$. $c(t)$ has duty cycle $d = 0.4$ with switching frequency of 100 kHz. Defaults:

$$V_{\text{offset}} = 0, D_{\text{min}} = 0, D_{\text{max}} = 0.9.$$



Gate Driver Block

The gate driver converts an input control signal $c(t)$ to a high-current output signal suitable for driving the gate of a power MOSFET transistor.

The input is a 0-5V logic signal relative to the input reference terminal

A power supply is required to operate the driver output (12 V in example at left)

The output is connected to the MOSFET gate, and the output reference is connected to the MOSFET source. The output high level is determined by the power supply. The driver level-shifts the signal, so that the input reference (ground) need not be the same as the output reference (MOSFET source)

Driver output voltage

Driver input voltage $c(t)$



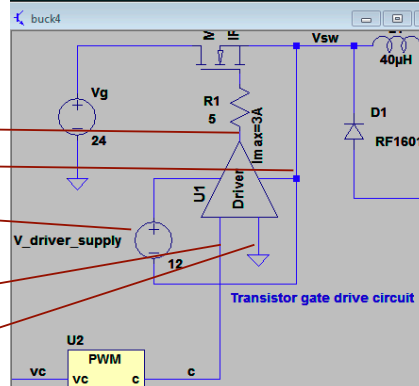
Output signal

Output reference

Driver power supply

Input signal

Input reference



LTspice Simulation of Buck Converter

MOSFET M1 and diode D1 operate as the switch

The converter input power source is $V_g = 24\text{ V}$

The converter output voltage V_{out} is applied to a load resistor $R_{load} = 5\text{ ohms}$.

The simulation models several sources of loss:

- Transistor and diode forward voltage drops

- Transistor and diode switching losses

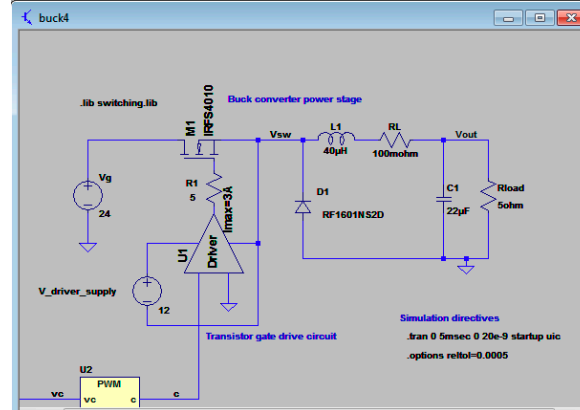
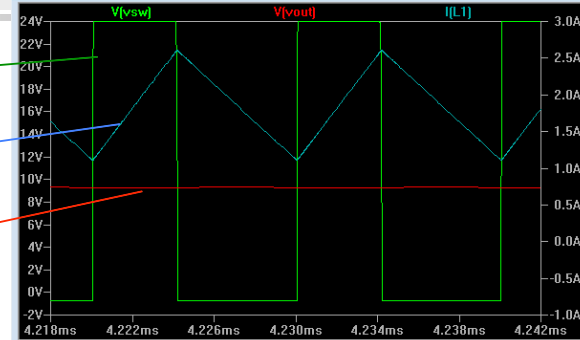
- Inductor winding resistance R_L

- Gate driver power consumption

Switch node voltage

Inductor current

Output voltage



Module 1 Homework Assignment

LTspice simulation of a Boost dc-dc converter

1. Download LTspice and boost simulation files onto your computer
2. Run simulation
3. Use LTspice to find steady-state output voltage, inductor current, and system efficiency