Iowa State University
Electrical and Computer Engineering
E E 452. Electric Machines and Power Electronic Drives

Laboratory #4
Introduction to dc/dc choppers

Summary

In this lab, PWM waveforms will be generated and analyzed. Additionally, a dc/dc converter (the first-quadrant buck chopper) will be simulated.

Learning objectives

- Understand basics of PWM waveforms.
- Design of a simple dc/dc chopper circuit.

Background material (should be read before coming to the lab)

- Trzynadlowski chapters 1 and 6.2.1
- Agilent 33210A function generator User's Guide – *especially the description of the PWM starting on page 84*
- MATLAB User's Guide – *in particular, writing simple M-files and functions*

Exercises and Questions

Instructions: every student should deliver his/her own report at the end of the lab session, even though the experiments are conducted in groups. You may want to answer the questions as you go along the exercises. Time yourselves according to the recommendations below.

1. Pre-lab assignment

Using paper and pencil, design a buck converter similar to that of Figure 1. You will verify your design in the lab, via simulation. Your converter must provide 50 V to a resistive load of R = 300 Ω. The input voltage is 100 V. You must select the switching frequency and inductor size so that the following specifications are met:

(i) Switching frequency < 10 kHz.
Remember that higher switching frequencies lead to higher losses in the semiconductor devices, so you should try to make this as small as possible.

(ii) The inductance must be one of the feasible values that may be obtained using equipment in the lab (single, series, or parallel combinations of 0.8 H, 1.6 H, or 3.2 H).
Remember that higher inductance means more weight and cost, so you should try to make this as small as possible.

(iii) The ripple factor of the output voltage should be less than 0.1.
After a certain frequency, the output voltage will resemble a triangular waveform. (Can you explain why?) Use a slightly modified version of equation (6.9) that uses voltages instead of currents to calculate the maximum voltage ripple that will satisfy the ripple factor constraint.

DELIVERABLE 1: Submit your written design, including all equations and calculations. What is the maximum voltage ripple allowed for the specified design constraint? What is the voltage ripple for your specific design?

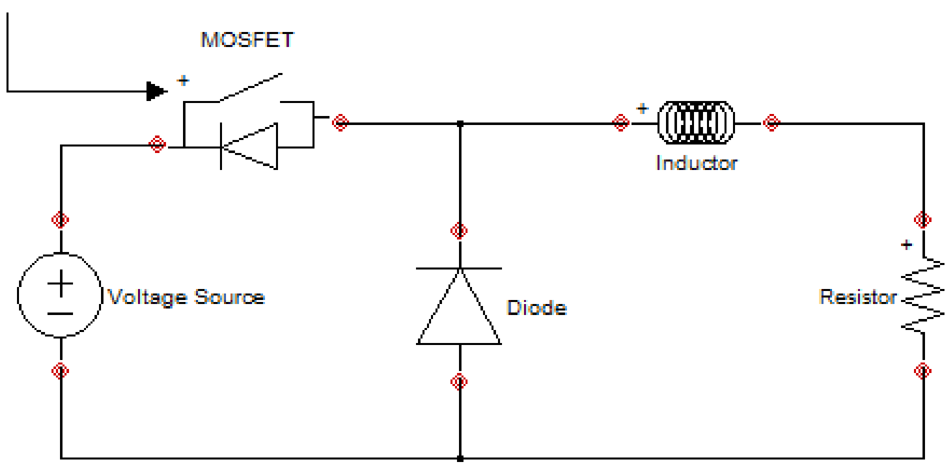

Figure 1. Buck converter circuit topology.

2. **Pulse-width modulation** [75 minutes]

Use the Agilent waveform generator to create a square-wave pulse-width-modulated (PWM) signal.

- Press "Pulse" and set the pulse period to 1 ms and the duty cycle to 50%. Set the peak-to-peak value of the pulses to be 1 V. Press "Graph" to see a graphical explanation of these parameters.

DELIVERABLE 2: Observe the waveform on the scope, and sketch it.

- Press "Mod" and set the modulating signal to a sine wave and its frequency to 40 Hz. Set the *Duty Cycle Deviation* parameter to 50%, so that the pulses range from 0% to 100% duty cycle. Press "Graph" to see a graphical explanation of the parameters.

DELIVERABLE 3: Observe the waveform on the scope, and sketch it.

- Save the data to a CSV file, and transfer to the PC.

- Use the FFT function of MATLAB to plot the amplitude spectrum of the PWM waveform. You may use the example code snippet below (borrowed from the MATLAB documentation file). First you have to load the CSV file in memory and store the PWM waveform as variable `y`.

```
T = t(2)-t(1);          % sample time
Fs = 1/T;               % Sampling frequency
L = length(y);          % Length of signal
t = t-t(1);             % Time vector (starts at t=0)

figure(1), clf
plot(t,y)

NFFT = 2^nextpow2(L); % Next power of 2 from length of y
Y = fft(y,NFFT)/L;
f = Fs/2*linspace(0,1,NFFT/2+1);

% Plot single-sided amplitude spectrum.
figure(2), clf
plot(f,2*abs(Y(1:NFFT/2+1)),'.-')
title('Single-Sided Amplitude Spectrum of y(t)')
xlabel('Frequency (Hz)')
ylabel('|Y(f)|')
```

DELIVERABLE 4: Sketch the amplitude spectrum. What do you observe?

3. **Buck Converter Simulation – Part I** [45 minutes]

   Use ASMG to simulate the buck converter you designed in the pre-lab. Make adjustments to the circuit as needed.

   DELIVERABLE 5: What is the parameter set that represents the best design? What characteristics make this design the best one? Make a sketch of the output voltage, output current, and input current.

4. **Buck Converter Simulation – Part II** [45 minutes]

   Add a filter capacitor in parallel with the resistive load. The capacitor value should correspond to a real component in the lab. Redesign the circuit using ASMG as a simulation tool, so that the same specifications are met.

   DELIVERABLE 6: What is the parameter set that represents the best design? Make a sketch of the output voltage, output current, and input current.

5. **Conclusion** [15 minutes]

   Write about one or two things you learned in this lab that you think are important or interesting, and why.