

Lecture 24

Computer Modeling and Simulation of PWM Converter Circuits

A.. Overview of Computer Simulation of Switched Converter Waveforms

1. General PWM Converter simulation goals
2. Special challenges to power electronic simulations
 - a. Non-linear switch conditions
 - b. Long time scale of the simulation
 - c. Need to Model Feedback loops
 - d. Case for Use of Simplified Models
 - e. Verification of Simulation with Experiment
3. A computer efficient simulation sequence is complex and artful
4. Two generic simulators for power electronics
 - a. Equation Solvers
 - b. Circuit simulators

B. Solid State Switch Models for PSPICE

1. **Overview**
2. Diode
3. MOSFET
4. IGBT

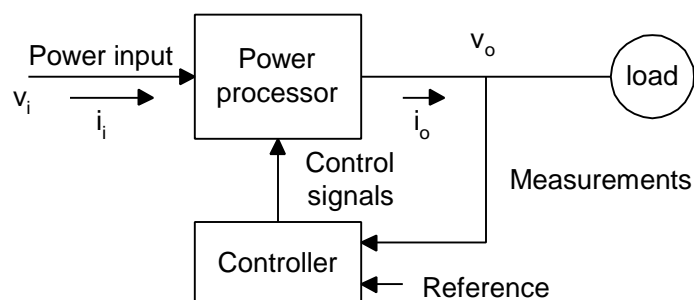
C. PSPICE BUCK PWM CONVERTER SIMULATION EXAMPLES

A. Overview of Computer Simulation of Switched Converter Waveforms/Circuits

Computer aided calculation programs like Mathematica (Wolfram Research) and Matlab (Mathworks) are useful in power electronics simulation. **But the most useful circuit waveform analysis is PSPICE (Microsim Corp.), as we shall see below.** The idea is to fully simulate the converter BEFORE starting construction and testing. Fixing problems in a model that we invest some time and effort in, may pay for themselves later by spotting problems before circuit construction begins. That is the promise of modeling, but not always the end result because of difficulties outlined herein.

1. Goals of Converter Operation Simulation

- a) Calculate both dynamic and steady state performances of the whole PWM converter system, for all voltages and currents versus time.



- Model power flow circuits
- Model controller response
- Model total system response with feedback loops included

- b) Obtain required V_{\max} and I_{\max} for all switching elements employed, so we can better specify required switches and **avoid switch failure** by better knowing the worst case switch stress conditions and planning for them.

- c) Estimate power loss in various components:

1. Switches are analyzed both from the two final DC switch states and the two **dynamic switching transitions**: from off to on and from on to off. Losses effect the heat generation and associated cooling requirements that we must meet.
- d). Magnetics Losses also contribute to heat flow requirements in addition to switch losses.
 - (a) Inductor's have core losses we can estimate
 - (b) Transformers also have core losses
- e) Snubbers and Control Electronics can be simulated to anticipate possible SOA problems of power switches.
- f) We can estimate thermal cooling for critical components using data from c and d above. Both solid state devices and magnetic cores want to remain below 100 °C to operate properly. So the cooling must result in ambient temperatures BELOW 100 °C in the converter. This must be addressed.
- g) We aim to speed-up converter and construction by COMPLEMENTARY simulation together with a proof-of-concept hardware prototype. We use both to iterate to a final design faster with fewer mistakes and a more reliable final product. That's the goal of simulation.

2. Challenges in Power Electronic Simulation

Unlike VLSI design we are simulating a discrete component system that has a variety of unique switching devices with MANY parasitic elements that often dominate PWM converter operation.

- a) Accurate non-linear switch models are not always easily available, even from the manufacturers. If available from the manufacturer's data sheets, the data may not always be as accurate as needed. Nevertheless, the WWW page of various manufacturers. e.g. Motorola, Harris,

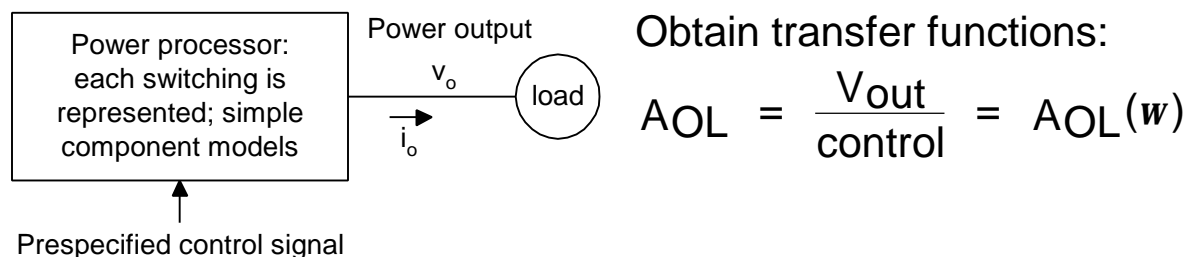
Siemens, IXYS, etc offers a wealth of information.

- b) The switching time t_{sw} is usually μsec but the converter system response time is seconds to minutes \Rightarrow **lots of computer time is required to simulate PWM converter waveforms well.**
- c) Chosen feedback controller models needs to be placed into the closed loop system model. Usually both current and voltage feedback loops are employed for PWM dc-dc converters. This complicates analysis.
- d) Early on we need to **choose proper simplification steps** so as to model only the immediate objectives. Limited simulation in early stages of design employs only a few switching cycles rather than millions or more to **crudely estimate voltages and currents, which repeat with each cycle.** This simplification step is very important. Often it must include only known to be dominant parasitic elements from prior experience! ! Include expected parasitic and circuit impedance's on the input and output as well as line inductance 100 - 500 μH for 208 ac mains. Also, include filter capacitors, snubbers, etc.
- e) Verify any model prediction by prototype PWM converter construction and testing.
- f) Include the control loop model and any surprises that may arise. Choose to model only portions of the power processor circuit rather than the full circuit in initial simulations to save time and simulation costs.

3. The converter simulation sequence is complex

The switch model replaces a time varying circuit topology with a single time invariant two port equivalent circuit by averaging over the switch cycle. Switch models are given by time independent averaged characteristics over T_s . Yet, the model changes with changing choice of duty cycle. Clearly, the frequency response of any such approximate model is limited to $f \leq f_{sw}$. The fact that we can even make an AC model of the PWM converter is surprising to some. Second semester we will make just such models by small signal expansion of the DC models we have made to date.

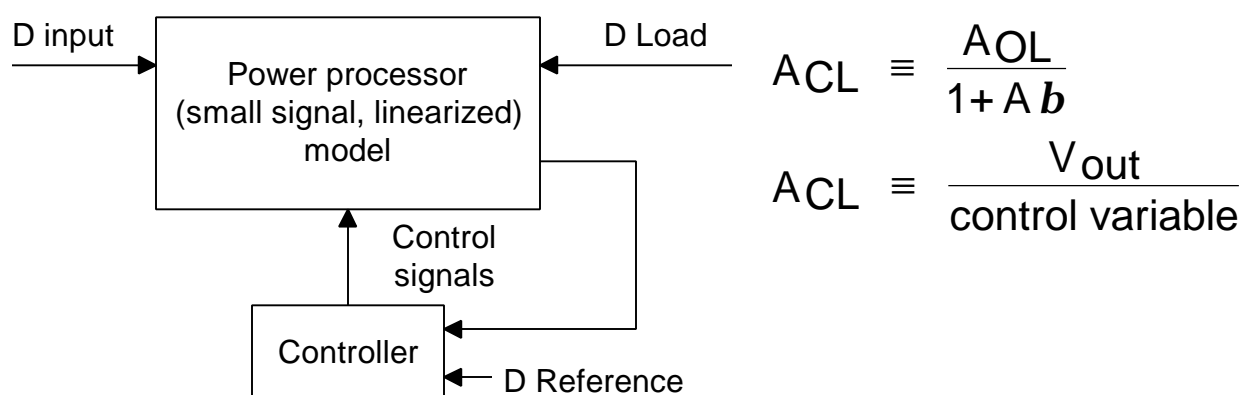
The $f \leq f_{sw}$ open loop power processor model $A(w)$ Results from such an exercise. For now just consider this small signal function as given:



In practice converters all employ feedback to stabilize the output to within the user's specifications. The chosen controller model employed is usually available from:

1. Control chip manufacturer
2. Controller software

The closed loop system for the controller is then:



One employs feedback because one expects big improvements in converter performance by using feedback including:

- Improved system stability to circuit component or switch device changes with both temperature and aging.
 - Higher Z_{in} , Lower Z_{out} for the closed loop system
 - A low frequency model allows the use of Bode plots for determining $A_{OL}(w)$. From open loop gain and phase plots versus frequency we can predict system stability as well as instability at all operating frequencies. One can also estimate system dynamical response to transient changes in the load or input by behavior of $A\beta$ the loop gain near $1 \angle 180^\circ$.
3. Two generic types of simulators are available for power electronics based simulators: Circuit oriented and differential equation based

a) Differential Equation Based Simulators

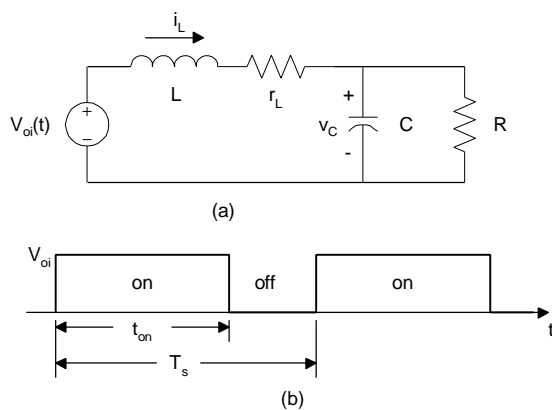
Give total control of the solution to the non-linear differential equations that describe the PWM circuit and then choose:

- The Integration method.

- Step time of simulation.
- First choose only those important terms in algebraic and differential equations. Neglect as many elements as possible for the first simulation to save computer time.
- Use C language to write the program steps.
- Choose your own graphical plotting tools.

b) Equation solver solution sequence

1. Circuit topology choice and switching sequence both determine loop equations we must solve.



$$r_L i_L + L \frac{di_L}{dt} + v_C = v_{oi} \text{ (KVL)}$$

$$i_L - C \frac{dv_C}{dt} - \frac{v_C}{R} = 0 \text{ (KCL)}$$

or matrix form

$$\begin{bmatrix} \frac{di_L}{dt} \\ \frac{dv_C}{dt} \end{bmatrix} \begin{bmatrix} -\frac{r_L}{L} & -\frac{1}{L} \\ \frac{1}{C} & -\frac{1}{CR} \end{bmatrix} \begin{bmatrix} i_L \\ v_C \end{bmatrix} + \begin{bmatrix} \frac{1}{L} \\ 0 \end{bmatrix} v_{oi}(t)$$

2. The State variable matrix format introduced in Erickson Chapter 7 is useful. For now consider the circuit equations just in the standard state variable or matrix form:

$$\frac{dx(t)}{dt} = Ax(t) + bg(t) \quad x(t) = \begin{bmatrix} i_L \\ v_C \end{bmatrix} \quad \text{and} \quad g(t) = v_{oi}$$

$$A = \begin{bmatrix} -\frac{r_L}{L} & -\frac{1}{L} \\ \frac{1}{C} & \frac{1}{CR} \end{bmatrix} \quad \text{and} \quad b = \begin{bmatrix} \frac{1}{L} \\ 0 \end{bmatrix}$$

3. The numerical solution: time step Δt , uses a linear interpolation

$$x(t) = x(t - \Delta t) + \int_{t - \Delta t}^t [A(z)x(z) + b(z)g(z)] dz$$

implicitly known are the small signal equations from which we can start the process of calculating the required integral

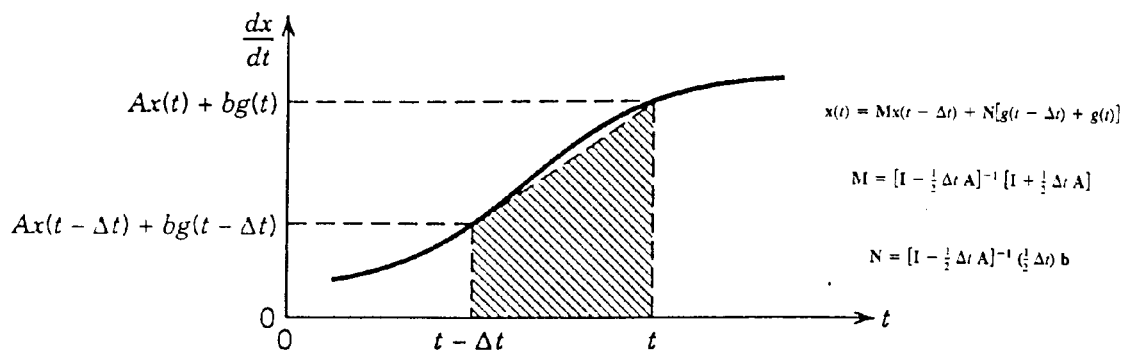
↓

↓

$$x(t) = x(t - \Delta t) + 1/2 \Delta t [A(t - \Delta t)x(t - \Delta t) + A(t)x(t)] \Rightarrow \text{Linear}$$

$$+ 1/2 \Delta t [b(t - \Delta t)g(t - \Delta t) + b(t)g(t)] \Rightarrow \text{Algebra}$$

Trapezoidal area approximation may be employed:



Use Matlab or Mathematica to do the numerical integration. You have complete freedom to employ other integration algorithms as well as to change the time sequences to suit your desire for a complete or just a crude model as we show below on page 9. The point to ponder is whether or not you have the interest in becoming a computer programmer of power electronics problems as this is a very time intensive endeavor.

Sample results of simple models are given on page 9 when using the Matlab simulator.


```

% Solution of the Circuit in Fig. 4-6 using Trapezoidal Method of Integration.
clc, clg, clear
% Input Data
Vd=8; L=5e-6; C=100e-6; rL=1e-3; R=1.0; fs=100e3; Vcontrol=0.75;
Ts=1/fs; tmax=50*Ts; deltat=Ts/50;
%
time= 0:deltat:tmax;
vst= time/Ts - fix(time/Ts);
voi= Vd * (Vcontrol > vst);
%
A=[-rL/L -1/L; 1/C -1/(C*R)];
b=[1/L 0]';
MN=inv(eye(2) - deltat/2 * A);
M=MN * (eye(2)+ deltat/2 * A);
N=MN * deltat/2 * b;
%
iL(1)=4.0; vC(1)=5.5;
timelength=length(time);
%
for k = 2:timelength
x = M * [iL(k-1) vC(k-1)]' + N * (voi(k) + voi(k-1));
iL(k) = x(1); vC(k) = x(2);
end
%
plot(time,iL,time,vC)
meta Example

```

Illustrative
lines of code
in Matlab

The results are plotted below for a buck converter, which will later to be modeled also by Pspice which is a circuit simulator

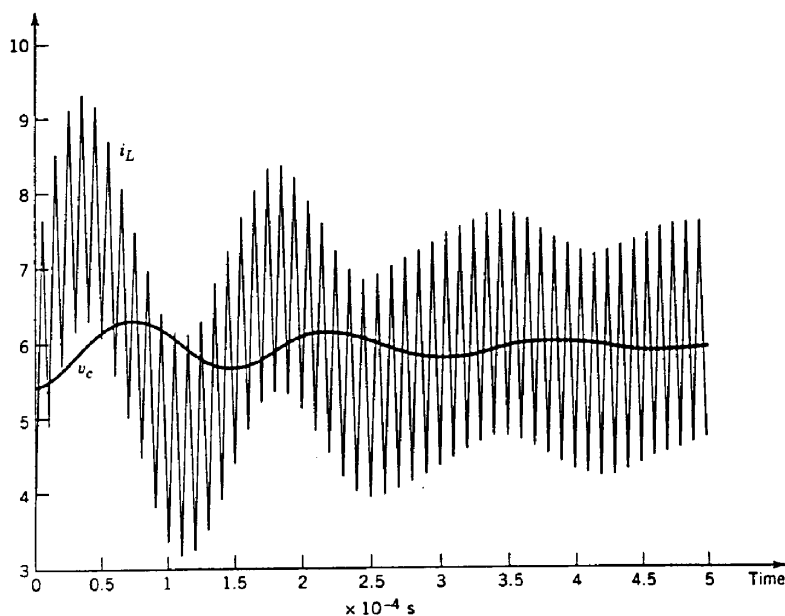


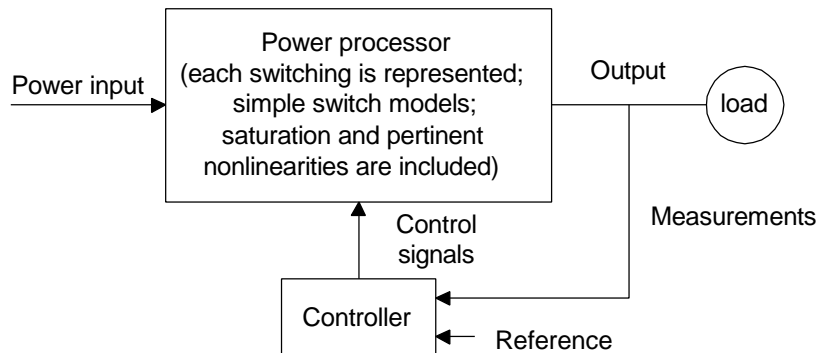
Figure 4-12 MATLAB simulation results.

With this equation based approach we see both the fast f_{sw} waveform and the slower averaged output waveforms for the PWM converter evolve during power up of the circuit. In this way we could estimate any peak excursions that might cause switch failure.

b) Circuit simulators

1) Overview

These simulators have been developed to a high level by the needs of VLSI chip design over the past decades. However, the IC environment is not the power electronics environment and we will have to take great care. A special form of commercial circuit simulator that we will favor is termed PSPICE.



The key is to break out the power processor portions from the controller portions.

You supply:

Circuit topology and nodes
Component values

Simulator generates:

Circuit equations and time evolving solutions transparent to the user for the power processor

2) Advantages of circuit simulators

- a. Fast/easy as prior library of circuits exists.
- b. Segmentation into modules is also easy.
 - (1) Basic building block PWM converter simulation modules exist. Test each separately then combine.
 - (2) Easy to change the design
 - (1) Vary Circuit topology itself
 - (2) Alter Component values and component models versus f , temperature, etc.

- 3) Disadvantages of circuit simulators
 - a. No easy control over
 - (1) Time step of simulation as for power electronics simulation times of seconds are required yet f_{sw} is giving $T_s \approx 10\text{-}100 \mu\text{s}$.
 - (2) No clear best choice of integration method for numerical solution to circuit equations
 - (3) Questionable assumptions of simulator model as these are general purpose tools but rather are often aimed at IC's and not always modified to power electronics.

- 4) Commercial simulators
 - a. SPICE and PSpICE (student freeware)

Simulation Program with IC Emphasis (SPICE) is widely available. It has easy to use shells and drop down menus to perform **what if circuit analysis** such as: DC and AC sweep analysis, transient response, parametric analysis of various device / component changes and finally temperature heating analysis. Two possible starting points with limited capability as compared to full-blown PSpICE are:

- 1) PSpICE version 7.1, MicroSim Corporation, 20 Fairbanks, Irvine, CA 92718. You can also get a student version from the www. Limited to 64 circuit nodes and 30 components.

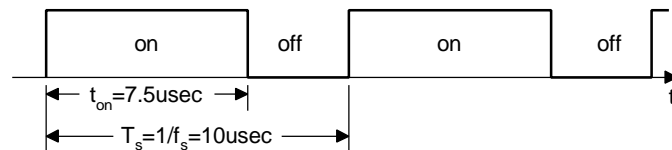
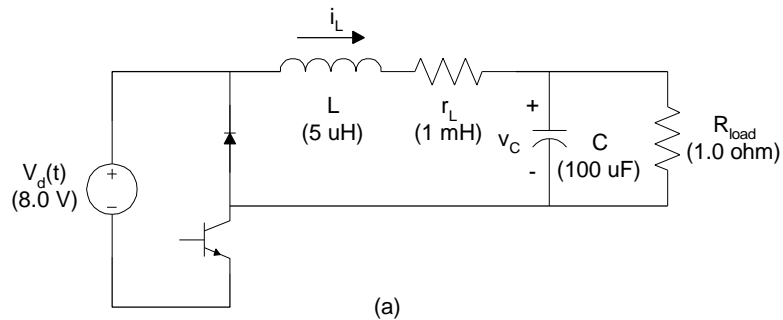
- 2) "Power Electronics: Computer Simulation, Analysis, and Education Using Evaluation Version of PSpice," on diskette with a manual, Minnesota Power Electronics, P.O. Box 14503, Minneapolis, MN 55414.

b) ILLUSTRATIVE PSPICE ANALYSIS

Circuit Schematic

Controller Input Data:

Unless specified switch device specs come from general purpose switch library which uses default values unless you change them to values for the specific power devices you are employing.



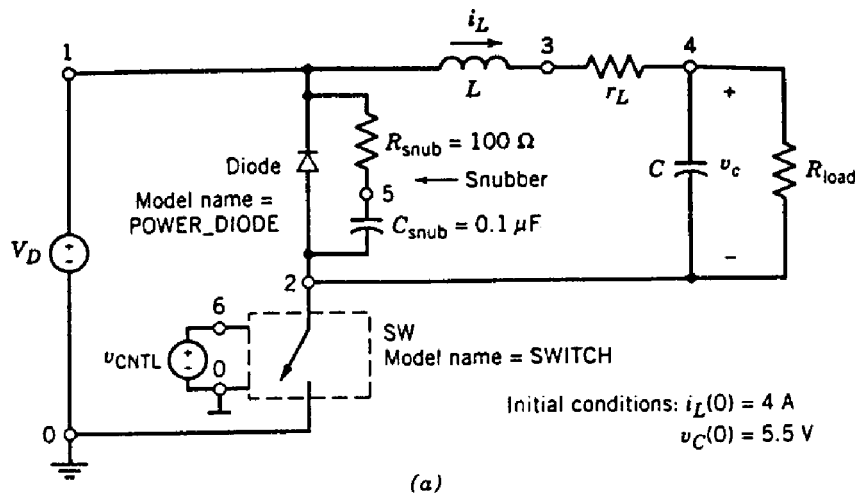
(a) Circuit for simulation. (b) Switch control waveform.

SPICE protocols must be followed:

- Assign node #'s with ground being always (0).
- A repetitive control signal is modeled in SPICE by utilizing V(control)
- Additional SPICE circuits must often be added to the switching nodes - like a diode R-L-C snubber - because discontinuities in SPICE stop the simulation.
- The more parasitic elements used make the simulation more realistic! In Power Electronics two-thirds of all components are not on the bill of material (Bruce Carstens). Also, the veracity of simulator results will improve.

One can create a schematic of the circuit and Pspice will automatically create a net list. To launch, PSPICE double click the Schematic icon. See details in the handout provided in class "Introduction to Pspice for Power Electronics" on disc and hard copy from G. Collins.

For example, the following is a SPICE simulation:



With PSPICE you can see the use of a R-C-diode snubber which softens V_D discontinuities caused by switching

PSpice Example

```

*
DIODE 2 1 POWER_DIODE
Rsnub 1 5 100.0
Csnub 5 2 0.1uF
*
SW 2 0 6 0 SWITCH
VCNTL 6 0 PULSE(0V,1V,0s,1ns,1ns,7.5us,10us)
*
L 1 3 SuH IC=4A
rL 3 4 1m
C 4 2 100uF IC=5.5V
RLOAD 4 2 1.0
*
VD 1 0 8.0V
*
.MODEL POWER_DIODE D(RS=0.01,CJO=10pF)
.MODEL SWITCH VSWITCH(ROn=0.01)
.TRAN 10us 500.0us 0s 0.2us uic
.PROBE
.END

```

(b)

Figure 4-9 PSpice simulation of circuit in Fig. 4-8.

SPICE Input file for circuit components with node assignments. This can be done automatically by schematic capture features of Pspice.

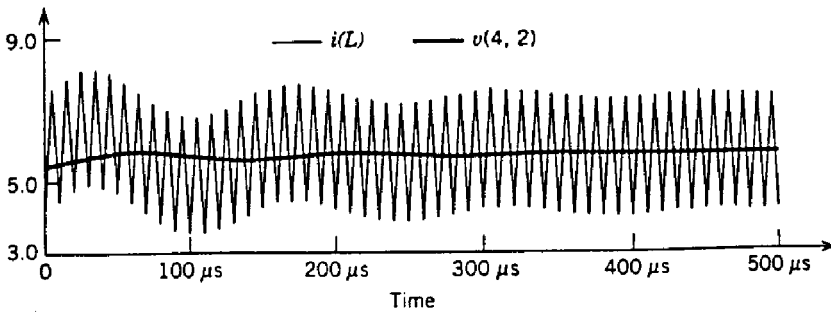


Figure 4-10 Results of PSpice simulation: i_L and v_c .

Typical output waveform from PSpice produced by a graphical post processor embedded within PSpice via Analysis / Set-up / Transient simulate menus.

However, PSpice is not optimized for the case of power electronic devices used as switches in the control of large amounts of power. Rather, it is for IC modeling with those types of devices for low power information technology found on logic chips. Perhaps a more focused power electronics simulator is PowerSim. However, it is not very robust nor easy to use.

Suggested Paper for 20 % of grade could employ PSpice, PowerSim, or other circuit simulators for buck-boost, Cuk, Flyback converter designs. The student version can be downloaded from the world wide web from: www.portal.ca/~powersim

For HW#4 Due in 1 week:

1. Answer any Questions asked in the class notes throughout lectures 20 -24.


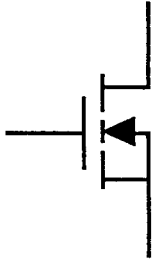
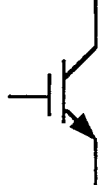
2. Erickson Chapter 4 Problems 2, 4, 5, and 6.

To finish out lecture 24, we review below proper models for the power electronic devices we employ in the second electronics revolution. Many device features are unique to power devices and are not familiar to engineers who work only in information technology devices

B. Solid State Switch Models

1. Overview

We will outline below only the full power electronic models for three switches that are the most popular. Other device models, such as thyristor models will be left for student papers for 20 % of the course grade. The three devices we choose are shown below:

1. Diode Models 
2. MOSFET Models 
3. IGBT Models 

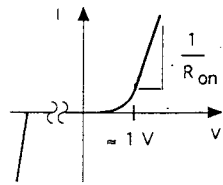
2. Power Diode Behavior and Model

The power diode has unique characteristics such as:

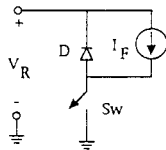
- A large forward recovery voltage V_{fp} that information technology diodes do not possess
- A finite time for the power diode to reach reverse recovery, that in information technology diodes is nearly instantaneous. The instantaneous reverse recovery will cause large reverse overvoltages that the power diode will not cause.
- Power diodes have unique quasi-static models as shown below on page 16.

PN Junction Diode Circuit Behavior

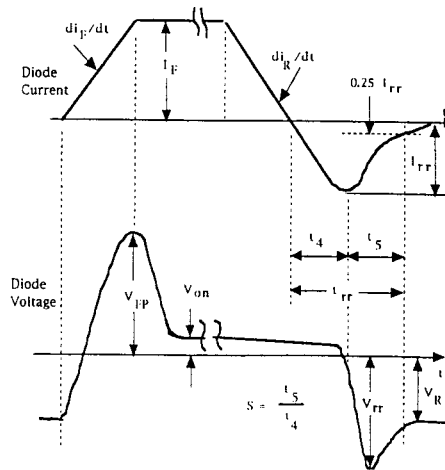
Static Characteristics



Typical Application



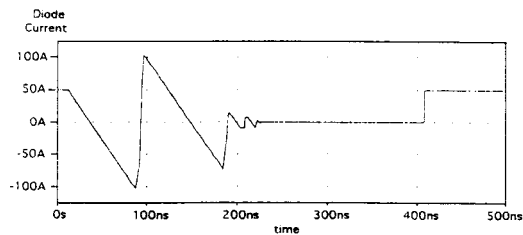
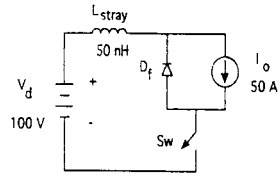
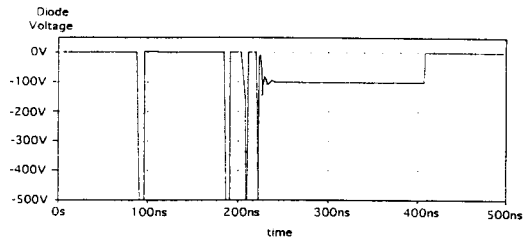
Switching Characteristics



Summary of PSpice Diode Model

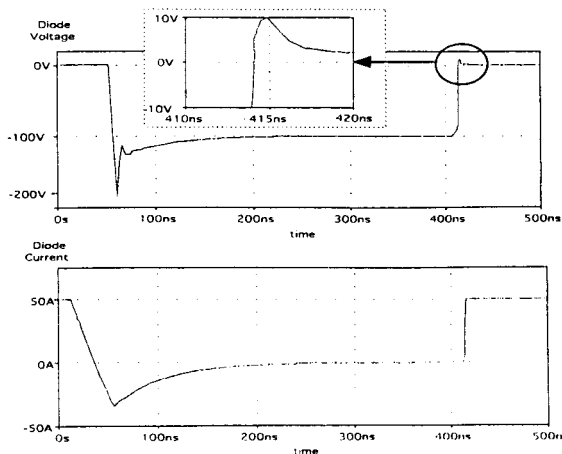
- Static I-V characteristic modeled with reasonable accuracy.
- Quasi-static approximation results in poor accuracy in modeling transient response of power diodes.
 - No forward recovery voltage V_{fp}
 - Instantaneous snap-off of reverse recovery current.
 - Very large reverse overvoltages due to instantaneous snap-off of reverse recovery current.

Simulation with PSpice Diode Model



- Step down converter test circuit.
- D_f modeled by built-in PSpice diode model.

Simulation Results with Improved Diode Model

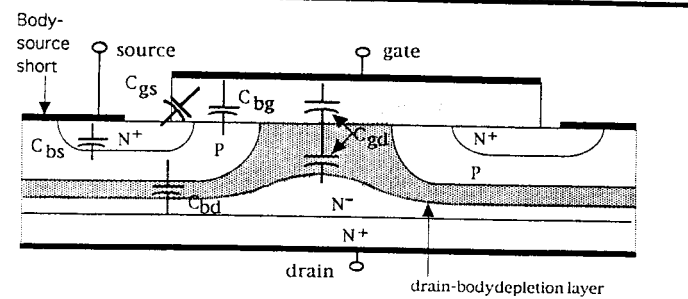


Models with the simple PSPICE information technology diode models will give very different simulation results from those obtained with the power diode models as shown above. Hence, PSPICE canned diode models are not to be used.

2 MOSFET

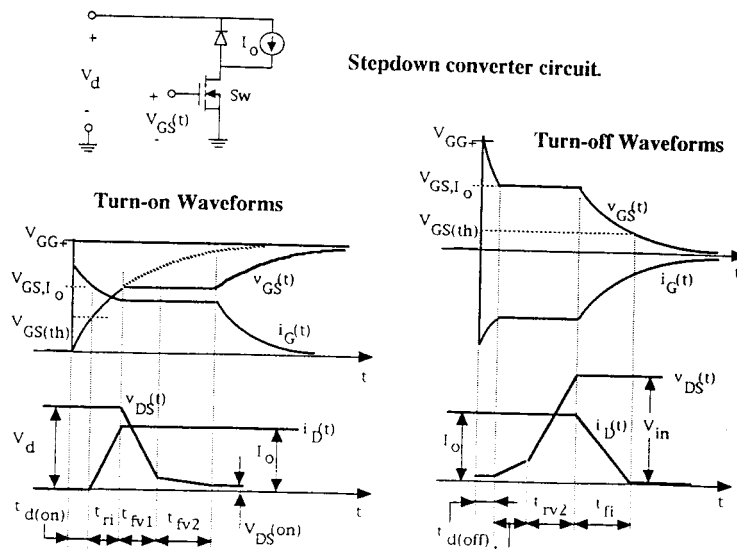
The power MOSFET models are also unique and the canned PSPICE information technology MOSFET models are inadequate

Voltage-Dependent Capacitances of Power MOSFETs



- Vertical geometry puts depletion layer in series with electrostatic component of C_{gd} .
- Large changes in depletion layer capacitance with changes in drain-source voltage. Hence 10 to 100:1 changes in C_{gd} measured in high voltage MOSFETs.
- Moderate changes in C_{gb} and C_{bs} .
- Circuit simulation models of MOSFETs must incorporate these large changes.

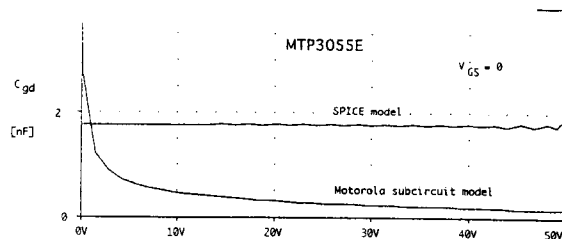
MOSFET Transient Behavior



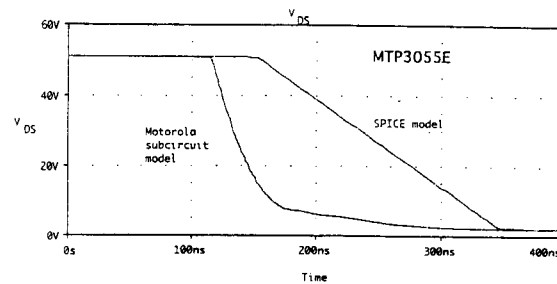
to the task of PWM converter simulation as shown on page 19 in comparative plots. We

compare the canned SPICE MOSFET information technology model to a Motorola power electronics MOSFET model that

Inadequacy of Built-in Pspice MOSFET Models



- Comparison of C_{gd} versus V_{DS} for PSpice MOSFET model and an improved model.



- Comparison of V_{DS} turn-off waveforms for PSpice MOSFET model and an improved model.

Summary of PSpice MOSFET Model

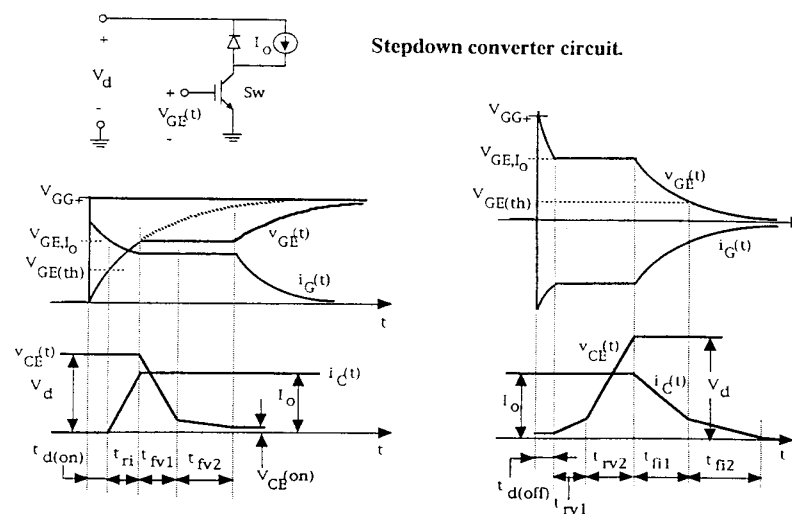
- Model gives reasonable simulations of static I-V characteristics.
- Does not model voltage-dependent gate-drain capacitance. Consequently transient responses of MOSFETs poorly simulated by built-in model.
- Better models needed which account for voltage-dependent gate-source and gate-drain capacitances.
- Several device manufacturers provide better models.

includes all the differences of note between **the two very different MOSFET devices.**

3. IGBT Power Devices

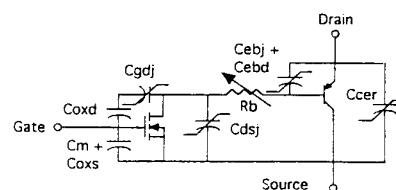
The built in SPICE IGBT model does not include the latest ultra-fast IGBT devices employing buffer layers, MV electron irradiation or free wheeling diodes. Still it gives decent results.

IGBT Transient Behavior



Parameter Estimation for PSpice IGBT Model

- Built-in IGBT model requires eight parameter values.
 - Parameters described in Help files of Parts utility program.
- Parts utility program guides users through parameter estimation process.
 - IGBT specification sheets provided by manufacturer provide sufficient information for general purpose simulations.
 - Detailed accurate simulations, for example device dissipation studies, may require the user to carefully characterize the selected IGBTs.

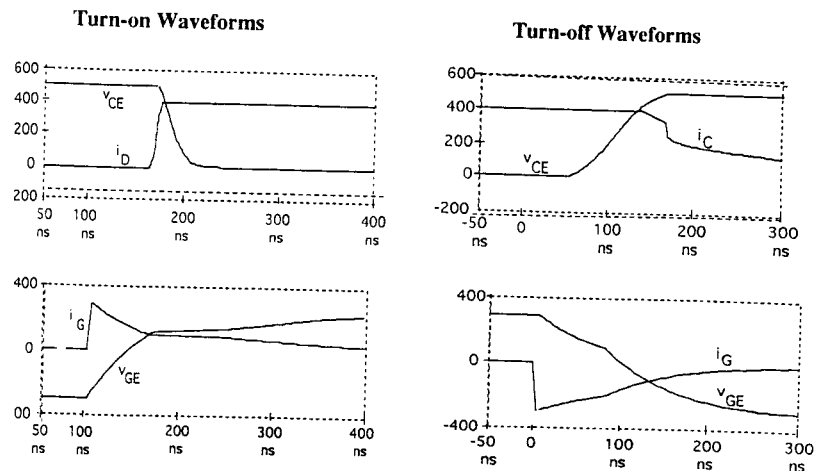


- Built-in model does not model ultrafast IGBTs with buffer layers (punch-through IGBTs) or reverse free-wheeling diodes

Simulation employing the canned SPICE model for an IGBT gives good results for a buck circuit as shown below.

Simulation Using Built-in IGBT Model

- Hard-switched step-down converter waveforms



Summary of PSpice IGBT Model

- Model provides reasonably accurate simulations.
- DC (static) characteristics modelled accurately.
- Transient behavior reproduces observed behavior. Quantitative accuracy adequate for most purposes.
- Model runs with reasonable speed - comparable with other built-in device models
- Parts parameter extraction utility program easy to use.
- Simulation results should be compared with experimental measurements.

4. Summary

Summary of PSpice Semiconductor Device Modeling Issues

- Built-in pn junction diode models inadequate for power electronics simulation. Incorrect modeling of reverse recovery.
- Built-in pn diode model can adequately simulate behavior of Schottky diodes.
- Built-in MOSFET model inadequate for simulating power MOSFETs. Model lacks voltage-variable gate-drain capacitance.
- Built-in IGBT model based on a proven model used in other simulators. Should provide reasonable simulation accuracy, but results should be checked experimentally.
- Several device manufacturers provide downloadable subcircuit models for selected devices which provide significantly better accuracy in simulation of transient behavior. These downloadable models are accessible from the manufacturers' web pages.

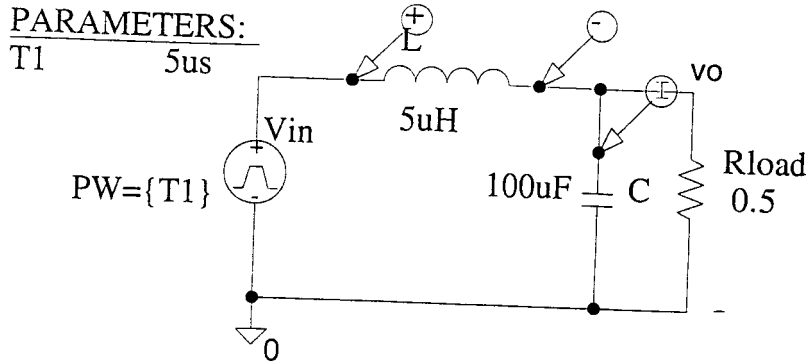
See the page 20 for a partial list of power device spec sheets from four manufacturers that are easily accessible from the WEB. The files are often in a PSPICE format for easy downloading to PSPICE.

Sources of Improved Power Semiconductor PSPICE Simulation Models

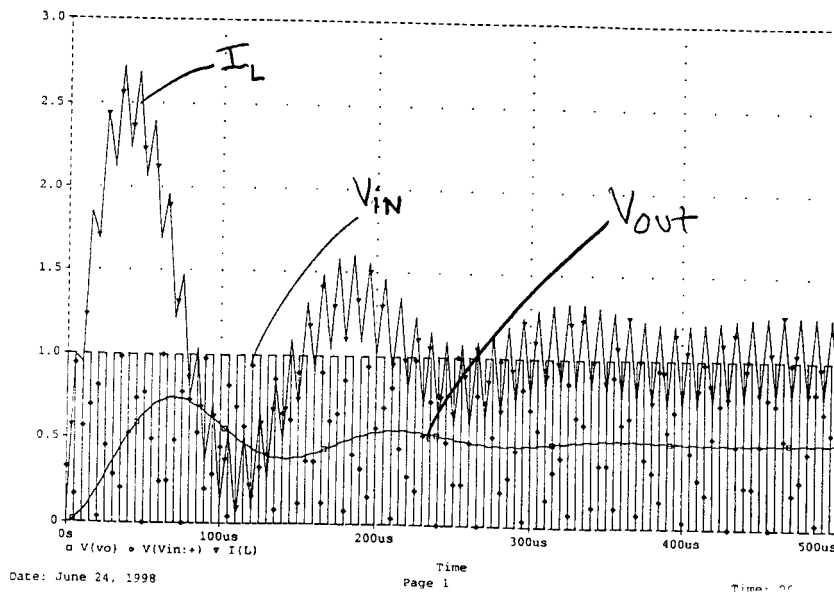
- Motorola
 - General WWW address: www.motorola.com
 - Specific address: <http://mot-sps.com/sps/General/chips-nav.html>
- Harris
 - General WWW address.: www.harris.com
 - Specific address: <http://www.semi.harris.com/families/models.htm>
- Siemens
 - General WWW address: www.siemens.com
 - Specific address: <http://w2.siemens.de/semiconductor/products/36/368.htm>
- IXYS
 - General address: www.ixys.com
 - Specific address: <http://www.ixys.com/pspice.html>

C. PSPICE BUCK PWM CONVERTER SIMULATION EXAMPLES

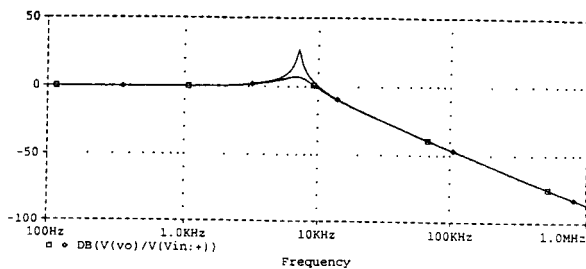
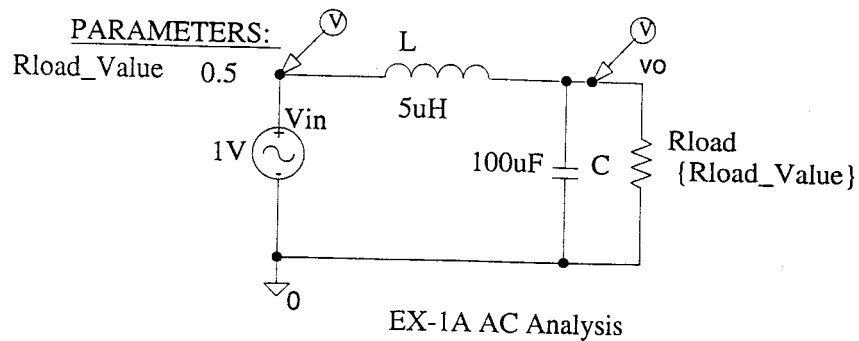
On the following pages we will first use PSPICE to solve for the waveforms in a simple BUCK converter without any feedback. I_L , V_{IN} , and V_{OUT} will be plotted versus time for the open loop circuit during start-up, showing BOTH the high frequency switch signal and the slowly varying components. We can also do frequency response plots of the open loop converter to capture the full open loop frequency response. Following these two plots we will then turn our attention to the BUCK converter with feedback added to see the effect of external disturbances on the circuit waveforms when feedback is employed. First we vary duty cycle to see changes in the BUCK waveforms with fixed V_{IN} . Then we vary the load R_L suddenly to change I_L in steady state but make V_{OUT} return to the value set by the feedback loop.



EX-1 Switch-Mode Principles

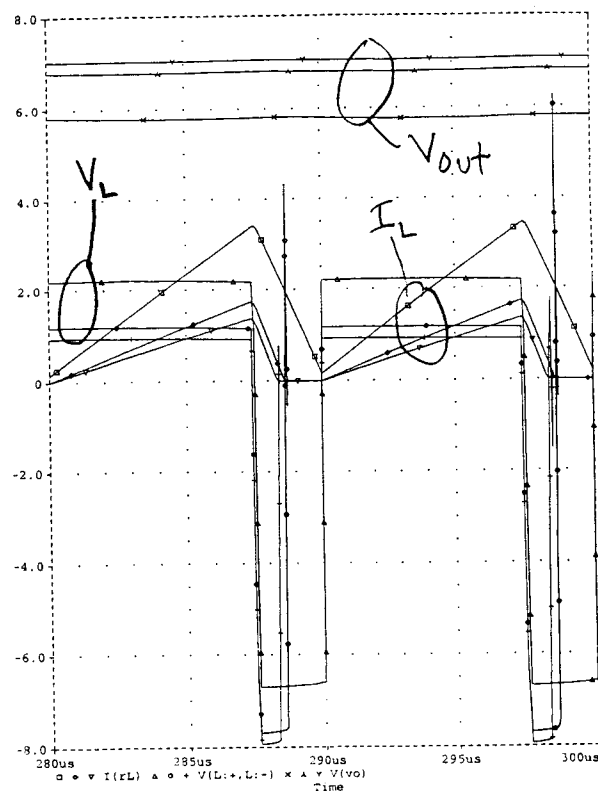
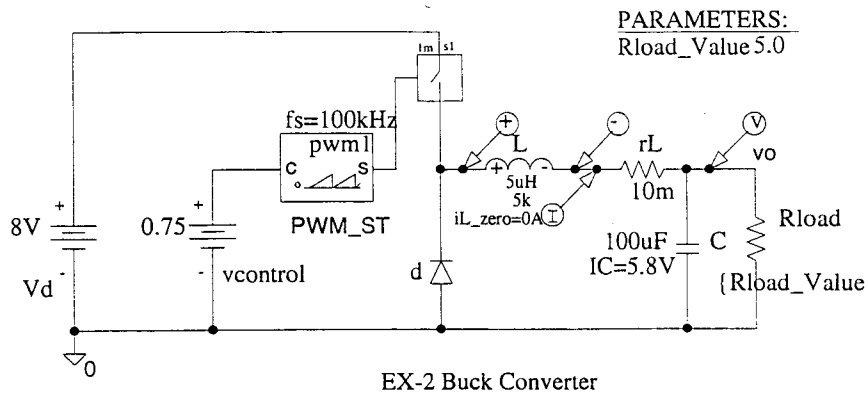


Above we see the start-up conditions of the PWM converter as it cycles towards equilibrium. Note that both I_L and V_{OUT} vary while the input voltage is constant as evidenced by the steady level of the input square-wave maximum level. From plots like this we can better determine any special switch stresses caused by circuit start-up. The frequency response of the BUCK circuit as simulated by PSPICE is shown on page 25. This plot is just a keystroke away when using PSPICE.

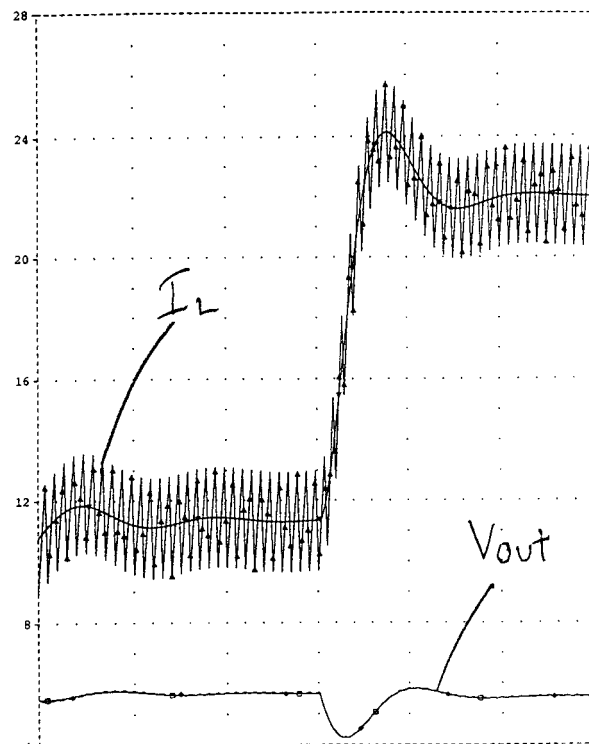
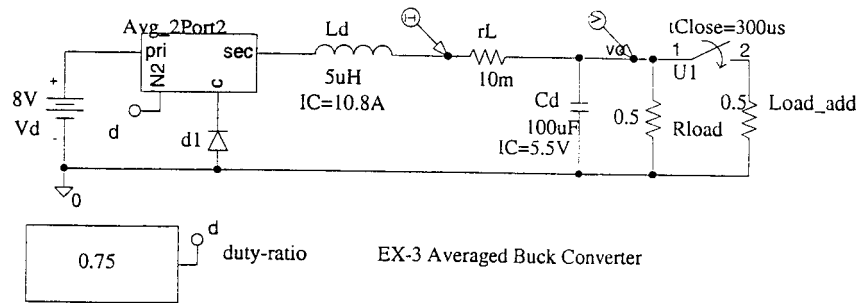


From such open loop plots one can employ BODE analysis to anticipate any possible instabilities when feedback is added to the BUCK PWM converter. On the next page we will add feedback to the BUCK circuit by employing a PWM control chip to change the duty cycle on the power switch according to the difference between the control signal and the output voltage feedback to the PWM chip. Note as the control voltage varies the duty cycle changes. Thus we find that V_L , I_L , and the output voltage all vary. Note that the magnitude of the change in voltage across the inductor varies as the duty cycle is decreased. On page 27 we plot the changes in the circuit waveforms of the feedback BUCK converter as the load changes suddenly. We are purposefully lowering the load impedance and increasing the load current, while trying to keep the output voltage steady by duty cycle control.

Peruse pages 26 and 27 to see the waveform changes caused by the external disturbances.



On page 27 we outline the waveforms for the case of a sudden change in the load impedance at 300 microseconds, causing increased load current and changing output voltage. The duty cycle readjusts to keep the output voltage constant and return to the nominal value set by the feedback.



For a term paper please use PSPICE to simulate some illustrative PWM converter circuits of your choice.