Instructions for LTspice Laboratory Exercise

# Introduction

This laboratory is divided into three parts that are meant to reinforce some concepts we have learned in class. The first part deals with simulations of linear circuits with just passive devices. There are several different ways to determine the circuit behavior we will find here, using either analytical or computer tools. We will use this section of the laboratory to learn about and familiarize ourselves with LTspice. In the second part of the laboratory, we will build a simple buck regulator switch mode power supply. This is an application for which SPICE, and therefore LTspice, is designed to handle, the analysis of circuits containing non-linear elements with complex voltage-current relations. In the final section of the laboratory, we will analyze details of the circuit involving power dissipation. When designing and building power supply circuitry we need to ensure that we use the elements within their proper operating ranges for voltages, currents, and temperatures. This section is also well suited for LTspice.

We note that simulations are just computer programs. Their accuracy depends on how well the LTspice models describe the behavior of the circuit elements and how accurately LTspice can calculate complex systems in a finite amount of time. Therefore, when we calculate different properties of the circuit, we may use different simulation configurations, trading some accuracy in less important calculations for execution speed to find the parameters of interest in a particular simulation.

# LC Filter

We consider a power supply that has a resistive load. We want to filter out the high frequency components from the power supply, so we construct an L-C low-pass filter (LPF) and place the resistor in parallel with the filter capacitor. We want to examine the properties of this circuit. The load resistor has a value of 5. The inductor in the LPF has a value of and the capacitor has a value of .

1. Open a new schematic: File->New Schematic (Ctrl-N)
2. Add components
   1. Voltage source: Edit->Component, P, or opamp symbol icon from toolbar
      1. Ensure top directory is C:\Users\*username*\AppData\Local\LTspice\lib\sym
      2. Select voltage, or type voltage in the search menu and select  
         or type V, or select voltage icon from toolbar
      3. Place and escape to exit
   2. Inductor: Edit->Inductor, type L, or select inductor icon from toolbar, place, Ctrl-R (three times) to rotate, escape to exit
   3. Capacitor: Edit->Capacitor, type C, or select capacitor icon from toolbar, place, escape to exit
   4. Resistor: Edit->Resistor, type R, or select resistor icon from toolbar, place, escape to exit
   5. Ground: Edit-> Place GND, type G, or select ground icon from toolbar, place, escape to exit
3. Connect components: Edit->Draw Wire, type W, or select Line icon from toolbar
   1. Click on GND, draw to V- and click
   2. Click on V+, draw in right angles to LHS of inductor, clicking on the corners and inductor
   3. Click on RHS of inductor, draw in right angles to top of resistor, clicking on corner of resistor
   4. Click on bottom of resistor, draw in right angles to bottom of line between GND and V-, clicking on corners and termination line
   5. Click on line from inductor to resistor and click on top of capacitor
   6. Click on line from resistor to ground and click on bottom of capacitor
   7. Escape to exit
4. Set the values of the components
   1. Inductor: Right click on the inductor, enter 500u in inductance box, OK
   2. Capacitor: Right click on the capacitor, enter 1000u in capacitance box, OK
   3. Resistor: Right click on the resistor, enter 5 in resistance box, OK
   4. Voltage source: Right click on the voltage source, go to Advanced, enter 0 in DC value box, enter 1 in AC Amplitude box, OK

## Frequency Response

We first find the frequency response of this circuit. The resonant frequency of the filter components is  
so we Simulate->Edit Simulation Command->AC Analysis and set up a Decade sweep with 1000 points per decade, starting at 10 Hz and ending at 10k Hz. This translates into a command  
.ac dec 1000 10 10k, which will appear on the LTspice schematic.

1. Setup the analysis
   1. Edit->Configure Spice Analysis or Simulate->Configure Analysis, type A, or select .ac icon on toolbar
      1. Select AC Analysis tab on Configure Analysis menu
         1. Type of sweep: Decade
         2. Number of points per decade: 1000
         3. Start frequency: 10
         4. Stop frequency: 10k
         5. OK
   2. Or select .t icon on toolbar, enter .ac dec 1000 10 10k in the text box, OK
   3. Click on schematic to place command
2. Save schematic as lcFilterFreq.asc in directory C:\files\ltSpiceFiles\homework
3. Run simulation: Either select Simulate->Run/Pause, type Alt-R on the schematic, or select the green triangle icon on the toolbar  
   (A netlist is created in lcFilterFreq.net. This contains the SPICE commands  
   \* C:\files\ltSpiceFiles\homework\lcFilterFreq.asc  
   V1 N001 0 0 AC 1  
   L1 N001 N002 500µ  
   C1 N002 0 1000µ  
   R1 N002 0 5  
   .ac dec 1000 10 10k  
   .backanno  
   .end  
   that tell SPICE about your schematic. Note that there are three voltage nodes, N001, N002, and 0 (GND), the four devices V1, L1, C1, and R1, the command for AC analysis (.ac directive), and an LTspice command, backanno, that allows you to select voltages and currents for graphical output. If I use the Place Netname command, either by typing N in the schematic or selecting the net icon from the toolbar, I can attach a label to, for example, the output and call it Vout. Unlabeled, LTspice identified this as N002. Labeled, the netlist now uses your defined label. Labeling can make the drawing more readable and also makes selecting values for plotting easier. In detail, I type N in the schematic, enter Vout in the text box, select the port type as Output, select OK, then place the port label and escape. The file lcFilterFreq.net now contains  
   \* C:\files\ltSpiceFiles\homework\lcFilterFreq.asc  
   V1 N001 0 0 AC 1  
   L1 N001 Vout 500µ  
   C1 Vout 0 1000µ  
   R1 Vout 0 5  
   .ac dec 1000 10 10k  
   .backanno  
   .end  
   The modified schematic, with the Vout label is shown in Figure 1)



Figure 1: lcFilterFreq.asc

1. Plots:
   1. Voltage: Hover over a wire with the cursor. If the wire is not at ground potential, a voltage probe will appear and the voltage node it is over will appear in the lower left corner of the LTspice window. The voltage on any node can be plotted by clicking on a wire connected to the node. A voltage difference can be plotted by first depressing the mouse key on one node, then sliding the probe over to a second node while the key is still depressed, and then releasing the key.
   2. Current: Hover over a device with the cursor. A current probe will appear and the device it is over will appear in the lower left corner of the LTspice window. The current through any device can be plotted by clicking on that device.
   3. Power: Depress the Alt key and hover over a device that can dissipate power (resistor or semiconductor). A thermometer will appear. Click and the power dissipated in that device will be plotted. (The thermometer icon to measure power only works for some types of LTspice simulations. It does not appear to work for .ac simulations. To plot the power in these cases, one can add an expression for the power as described in subsection d., below.)
   4. Functions of basic plots: Sometimes one wants to plot an arithmetic function of node voltages and/or branch currents. To do this, make the plot window (.raw) the active window and then add a trace by opening another window either typing A on the plot window or Plot Settings->Add trace, and then entering the mathematical expression in the “Expression(s) to add” box. For example, another way to add the voltage across the inductor, enter V(n001) – V(n002), OK, or to calculate the power through the damping resistor R1 would be to enter V(vout)\*I(R1), OK, where vout is the node on the high side of R1. No other node needs to be added since the other node is defined as the ground node.
   5. Cursors: The waveform plotting routine has two cursors. One can activate the cursor by left clicking on the label of a plot. A cursor box will appear with the horizontal and vertical coordinates of the waveform point under the cursor. Alternatively, one can right click on the desired plot label and select the cursors from the Attached Cursor box at the top of the Expression Editor. The choices are none, 1st, 2nd, or 1st & 2nd.
   6. One can change parameters of the plot by hovering the cursor along an axis outside of a plot until a ruler icon appears and then right clicking. That can allow you to change the axis values and, if a vertical axis, delete a family of plots, such as amplitude or phase for a Bode plot.
2. Run the simulation and then place a voltage probe on the resistor and view the amplitude and phase of the result. The peak of the response is close to (it is a little lower than the calculated value because of the effect of the resistance).
3. The phase of the response changes from to as the frequency goes through the resonance.

a. Record the frequency at which the response is a maximum and record the amplitude, in dB, at that frequency

\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_ Hz \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_ dB

1. Record the frequencies and amplitudes at which the phases are , , and , respectively  
     
   -45O \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_Hz \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_dB  
     
   -90O \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_Hz \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_dB  
     
   -135O \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_Hz \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_dB  
     
   This shows the maximum amplitude occurs at approximately -90O and that the amplitudes are down approximately 3dB at 45 degrees on either side of the 90 degree phase shift.
2. You can document your work for publication using Tools->Write image to .emf file. First select the window you want to convert to .emf and then use the Tools->Write image to .emf file command. This works for both the schematic and the data plots.

## Step Response

We now find the time response of this signal to a step impulse.

1. Delete (type Backspace or Delete, Edit->Delete, or X icon) the .ac directive from the schematic
2. Right click on the voltage source, already in Advanced, enter 1 in DC value box, and either enter 0 in AC Amplitude box or delete the 1 from the box to leave it blank, OK.
3. Type A, or Edit->Configure Spice Analysis or Simulate->Configure Analysis, or select .ac icon from toolbar -> Transient
   1. Stop time: 50m
   2. Time to start saving data: 0
   3. Select “Start external DC supply voltages at 0V”
   4. OK
4. Or enter Transient command: .tran 0 50m 0 startup, then OK
5. Click to enter command on schematic
6. Save file as lcFilterStep
7. Run simulation: Alt-R or green triangular icon, or Simulate->Run/Pause
8. Place a voltage probe on R1
9. Observe the ringing of the voltage on the resistor
10. Sketch the output voltage waveform over 50ms. Estimate and record the time between the initial peaks. To what frequency does this correspond?
11. Record the ratio of the amplitude of the second peak, with respect to the final value of the voltage, , to the amplitude of the first peak, with respect to . A single pole damped circuit decays exponentially. If the excitation were an impulse, the oscillation would be about the origin. If the excitation is a step, the voltage oscillates about the value of the step. If there is damping in the circuit, after a sufficiently long time the voltage damps to the value of the input step. Using the ratio of these peaks and the time difference between the peaks, calculate a damping time using the formula to fit the data.

Since the value of is the same at the two measurement times, solving for τ we get

Ratio: \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_

Damping time : \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_



Figure 2: lcFilterStep.asc

# Damped Filter

In order to damp the frequency response of our L-C filter, we add a damping resistor to the circuit. We will use LTspice to see what value of damping resistor to add in parallel with our load resistor.



## Parametric Analysis

In order to efficiently select the filter we will use the LTspice .STEP function that allows us to perform a parametric analysis. This introductory section gives an overview of how to perform such an analysis on the resistor in a damped filter. You will be stepped through the instructions later in this section of the laboratory exercise.

Vary the parametric component by right clicking on it. Under Component *Properties* *(Resistor, capacitor, or Inductor)* enter **{R}, or {C}, or {L}** in the *Resistance, Capacitance, or Inductance* box. Note that the braces **{}** around the parameter name are required. This tells LTspice the component value is a global parameter. Close the component box by selecting the *OK* button.

Add a spice directive to the circuit in one of three ways. Enter . on the schematic, use the *.t* command on the toolbar, or select the *SPICE Directive* from the *Edit* pull down menu. All of these ways open a dialog box *Edit Text on the Schematic*. Ensure the *SPICE Directive* radio button is highlighted. Type the needed command (see the example below) then left click the directive onto the circuit diagram.

For example, the command *.step dec param R 0.01 1.0 4,* steps the parameter **R** logarithmically from 0.01 ohms to 1.0 ohms in four steps per decade when the *Run* command is chosen. Display the plots for all of the steps by then selecting the voltage or current of interest and displaying the plot. To select only specific plots, choose *Select Steps* to display the traces of interest from the *Plot Settings* drop down menu.

In order to step through the parameter in linear increments, we modify the command to *.step param R 0.05 0.15 0.02,* which starts with **R** at 0.05 ohms, and steps in increments of 0.02 ohms to 0.15 ohms.

To return to the original value we can either delete the *.step* command and replace it with a command used to set **R** *.param R 0.1,* or remove the command altogether, to return to the damping resistor properties and replace **{R}** with the desired value.

This concludes the introduction.

# Parametric Analysis

1. Either work with the existing schematic, or load a previous schematic, File->Open lpFilterFreq.asc
2. Add an additional shunt resistor for extra damping: type R on the schematic, select the resistor icon from the toolbar or Edit->Resistor or select from menu, place, escape to exit
3. Connect resistor to existing circuit by typing W in the schematic, selecting the wire icon from the toolbar, or using Edit->Draw Wire. Escape when finished.

Set this second resistor as a parameterized object: Right click on the resistor and enter the resistance as {Rdamp} to tell LTspice that this resistance will be treated as a parameterized value.  
Hint: You need a starting value for the parameterized damping resistor. In an circuit, the only quantity you can create with the units of impedance is . Although this may not be the value of the “best” damping resistor, it should be close to that value, so is a good value to include in your parameterization.

Calculate and record \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_

1. Delete the .tran directive. Add the SPICE command to tell LTspice the parameters of interest: Edit->SPICE Directive  
   .step dec param Rdamp 0.01 10 4  
   This tells LTspice that Rdamp will be a parameter, where the value of R will be varied logarithmically from to with four steps per decade.
2. Set up an AC analysis from 20 Hz to 2 kHz:
   1. Type A on the schematic, select .ac from the toolbar, or Edit->Configure SPICE Analysis, or Simulate->Configure Analysis
      1. Type of sweep: Decade
      2. Number of points per decade: 1000
      3. Start frequency: 20
      4. Stop frequency: 2k
      5. OK
   2. Click on schematic to place command
   3. Ensure voltage source is prepared for the AC analysis. Right click on the voltage source, go to Advanced, enter 0 in DC value box, enter 1 in AC Amplitude box, OK
3. Save file as lcFilterDampFreq
4. Run simulation: Type Alt-R or select green triangle on toolbar
5. With the waveform window highlighted, either type Shift-S or select Plot Settings->Select Steps to select which of the parameterization values of Rdamp you wish to plot
6. Replace the .step command with a command that will vary the parameter linearly from to in steps of . Right click on the .step command to bring up the .step Statement Editor:
   1. Either use the prompts
      1. Name of parameter to sweep: Rdamp
      2. Nature of sweep: Linear
      3. Start value: 0.5
      4. Stop value: 2.0
      5. Increment: 0.1
   2. Or enter .step param Rdamp 0.5 2.0 0.05 in the bottom box
   3. OK
7. Run simulation: Type Alt-R, select green triangle icon on toolbar, or Simulate->Run/Pause
8. Choose the desired resistor value (when the output voltage waveform is slightly underdamped). Note that this “desired” resistor value is somewhat subjective. There are tradeoffs between such circuit parameters as rise time, overshoot, etc., that are determined by the choice of the resistor value. You should briefly explain why you chose your desired resistor value.
9. Replace the Resistance value **{R}** with the desired value and record this value

\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_

1. Delete the .step Spice command from the circuit. Run an AC simulation to verify this resistor value has the desired response and sketch the confirmation



Figure 3: lcFilterDampFreq.asc

## Transient Analysis

Leaving the circuit alone, modify the excitation to obtain a step response

1. Right click on the voltage source to set the source to a DC value
   1. DC value: 1
   2. AC Amplitude: 0
   3. OK
2. Right click on the .ac command to open the Edit Simulation Command box and
   1. Either
      1. Select Transient tab
      2. Stop time: 10m
      3. Time to start saving data: 0
      4. Select Start external DC supply voltages at 0V
   2. Or in lower box enter
      1. .tran 0 10m 0 startup
   3. OK
3. Save file as lcFilterDampStep.asc
4. Run simulation: Type Alt-R, select green triangle icon on toolbar, or Simulate->Run/Pause
5. Sketch the circuit output voltage. Sketch the power dissipated in the resistor (see sections 4.c and 4.d in the discussion of Frequency Response - Plots above).
6. Record the average and RMS voltage of the waveform and record the peak power, the steady state power, and the average power in the damping resistor, Rdamp. To obtain these calculations:
   1. Use the mouse to zoom over the entire window to select the entire plotted waveform
   2. Move your mouse to the label at the top of the graph for the waveform of interest and Ctrl Left Mouse Click to bring up the menu.
      1. For a voltage waveform, the box will show Interval Start, Interval End, Average (V) and RMS (V)
      2. For a power waveform, the box will show Interval Start, Interval End, Average (W) and Integral, the energy dissipated over the waveform, (J)
   3. Output Voltage Waveform
   4. Rdamp Dissipated Power Waveform



Figure 4: lcFilterDampStep.asc

# Praeg Filter

The circuit we just designed has the desired frequency response, but at the expense of dissipating most of the power in the damping resistor. If we mainly want to drive a DC load, we can insert a capacitor in series with our damping resistor to block the DC current and reduce the power dissipated in this resistor.



## Parametric Analysis

1. Load the lpFilterDampFreq.asc: File->Open lpFilterDampFreq.asc
2. If not already done, set up an AC analysis from 20 Hz to 2 kHz:
   1. Type A on the schematic, select .ac from the toolbar, or Edit->Configure SPICE Analysis, or Simulate->Configure Analysis
      1. Type of sweep: Decade
      2. Number of points per decade: 1000
      3. Start frequency: 20
      4. Stop frequency: 2k
      5. OK
   2. Or type . of select .t icon on the toolbar and enter .ac dec 1000 20 2k, OK
   3. Click on schematic to place command
3. Delete the .step statement for the parametric resistance sweep: Delete, select, escape
4. Modify the damping resistor (Rdamp.) parameter to include the desired resistance: Right click on resistor, replace {R} with desired value, OK.
5. Modify the circuit layout, if necessary, to allow a capacitor in series with Rdamp. M is Move, S is Stretch. Stretch is like move, but wires rubber band along with selected objects.
6. Delete the wire connecting Rdamp to ground: Delete, select, escape
7. Add a capacitor: Type C, select the capacitor icon from the toolbar, or Edit->Capacitor then place, escape to exit
8. Connect the capacitor to Rdamp and GND using w, the wire icon, or Edit->Draw Wire, escape to exit
9. Set C2 to be parameterized
   1. Right click on C and set Capacitance to {CPraeg}
   2. Add the SPICE command to tell LTspice the parameters of interest: Type .  
      .step param CPraeg 3000u 7000u 1000u  
      OK then place  
      to step through five values of the capacitance.
10. Add the node name (N) Vpraeg to the top of the capacitor CPraeg for plotting purposes
11. Save the file as praegFreq.asc: File->Save As praegFreq.asc
12. Run simulation: Alt-R or green triangle
13. Select and record an acceptable capacitor value for this load  
    Hint: From class notes

\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_ F



Figure 5: Praeg filter frequency sweep schematic

## AC Analysis

Perform an AC analysis to obtain the frequency response .ac dec 1000 20 2kHz

1. Record the frequency at the peak of the response and the relative gain (in dB) at this frequency

\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_ Hz, \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_dB

1. Compare these results with those from the damped filter.



Figure 6: praegFreq.asc

## Transient Analysis

Leaving the circuit alone, modify the excitation to obtain a step response

1. Right click on the voltage source to set the source to a DC value
   1. DC value: 1
   2. AC Amplitude: 0
   3. OK
2. Right click on the .ac command to open the Edit Simulation Command box and
   1. Either
      1. Stop time: 10m
      2. Time to start saving data: 0
      3. Select Start external DC supply voltages at 0V
   2. Or in lower box enter
      1. .tran 0 10m 0 startup
   3. OK
3. Select desired capacitance and replace the parameter C with that value. Delete the .step statement.
4. Save file as praegStep.asc
5. Run simulation: Alt-R of green triangle
6. Record the peak voltage, and the time of the voltage peak, and the average and RMS voltage of the waveform
7. Record the peak power, and the time of the peak power, the steady state power, and the average power in the damping resistor, Rdamp. Note that now there is no the steady state power dissipated in the damping resistor.
8. Output Voltage Waveform
9. Observe the currents on the filter inductor and damping resistor and the voltage on the Praeg capacitor and discuss their behavior with respect to that of the output voltage waveform, including their importance at different parts of the input step.
10. Rdamp Dissipated Power Waveform



Figure 7: praegStep.asc

# Buck Regulator

Now that we have the desired load and filter, we will build a buck regulator power supply and look at its step responses into the load.

## Initial Response

1. Load praegStep.asc: File->Open praegStep.asc
2. Ensure that the parameters for R2 and C2 have been replaced by the desired components and delete (delete or backspace) any unneeded step statements
3. Stretch (S) the filter components to the right of the schematic to make room for other components. Left click when the components are in the desired location, then escape.
4. Add components:
   1. IGBT:
      1. Type P or component icon from the toolbar or Edit->Component
      2. In top line, pull down menu and select custom directory (C:\files\ltSpiceFiles\sym)
      3. Select NIGBTikw75n65eh5 (650V, 75A),click OK
      4. Rotate the device (Ctrl-R) until the gate faces down
      5. Place between the voltage source and the inductor, escape to exit
   2. Voltage source:
      1. Select the voltage source icon from the toolbar
      2. Place to the right hand side of the existing schematic and escape to exit
   3. Diode:
      1. Type P or select the diode icon from the toolbar
      2. Select diode, OK, rotate (Ctrl-R) until the cathode faces up
      3. Place and escape to exit
   4. Resistor: Type R or select the resistor icon from the toolbar, place above the second voltage source V2, escape to exit
5. Modify schematic
   1. Delete (delete or backspace) trace between V1 and L1, escape to exit
   2. Move (M) the IGBT between the voltage source and the inductor
   3. M the diode so that it is in shunt with the voltage source and IGBT
   4. Draw Wire (W) to connect the voltage source V1 to the IGBT and then the inductor (You can drag one wire from start to end. LTspice is smart enough to skip over components as you drag the wire)
   5. Wire (W) a connection from the right of the R3 V2 series stack, through the stack, and then out to the right again.
6. Create and assign labels to the drawing. To make the drawing less cluttered, we will separate the gate driver circuit from the IGBT on the schematic. We will use label nets to make the connection
   1. Type N, select the net icon from the toolbar, or Edit->Label Net to create label igbtG.
   2. Place one at the end of the wire connecting to R3, place another at the gate of the IGBT, escape.
   3. N to create label igbtE, place one at the end of the wire connecting to V2, place another at the emitter of the IGBT, escape.
   4. To make the task of drawing the waveforms easier, create another label, igbtC, and place it at the IGBT collector, escape.
7. Set the remaining component values
   1. Set V1 to 300 VDC: Right click on V1, DC value 300, AC Amplitude 0, OK
   2. Set R3 to : Right click on R3, Resistance 10, OK
   3. Set D1 to carry 75 A with a breakdown voltage of 600 VDC
      1. Right click on D1
      2. Pick New Diode
      3. Select VS-E5PH7506
      4. OK
   4. Set V2 to be the gate drive, a pulse train to give a pulse-width modulated control of the IGBT. From the data sheet, a gate value of -5 V is guaranteed to keep the IGBT off and a gate value of 20 V to turn the IGBT on. We will start the pulse train immediately (Tdelay = 0), set the pulse period for , a clock, set its duty cycle to 50% (Ton = Tperiod/2), and set some fast rise and fall times of the gate drive pulser. Based on our filter response we expect the filter to settle out within a few ms, we set the number of cycles to 200 for a 20 ms excitation time. We will see the command  
      PULSE(-5 20 0 10.0n 10.0n 50u 100u 200) by V2
      1. Right click on V2
      2. Advanced
      3. Select PULSE function
      4. Vinitial = -5; Von = 20; Tdelay = 0; Trise = 10.0n; Tfall = 10.0n; Ton = 50u; Tperiod = 100u; Ncycles = 200
      5. OK
8. Set up a transient simulation:
   1. Type A or select .ac icon or, if it exists, edit existing .tran command
      1. Transient
      2. Stop time: 20m (this is NcyclesTperiod)
      3. Time to start saving data: 0
      4. OK
   2. This gives a simulation command of .tran 0 20m 0 startup
9. Save file File->Save As buckInitialIgbt.asc
10. Run simulation: Alt-R or green triangle icon on toolbar



Figure 8: IGBT buck regulator schematic

### Voltage and Current Analysis

1. Sketch , , on one plot, labeling times, voltages, and currents
2. Sketch , the current in the main Praeg capacitor C2 over 20ms and then over two cycles under steady-state conditions, after 19 ms. Include the values of the current on the plot.
3. On one plot sketch two cycles of the filter inductor L1 voltage (V(igbtE)-V(vout)), and current (IL1) under steady state conditions, after 19 ms. Include the values of the voltage and current on the plot. What is the relationship between the voltage across the inductor and the current through it? What is the average voltage across the inductor at that time?
4. On one plot sketch two cycles of the free-wheeling diode VD1, and ID1 under steady state conditions. Include the “on” and “off” values of the diode voltage. What is the nominal voltage drop across the diode when it is on?
5. On one plot sketch two cycles of the drain-to-source IGBT voltage (V(igbtc)-V(igbte)) and current Ix(U1:C) under steady state conditions. Include the “on” and “off” values of the IGBT voltage and the value of the current during the cycle. The voltage drop across an IGBT in the “on” state can be written as a linear equation . From the data, calculate and .

### Power Analysis

In a circuit design we must also ensure that we do not violate any power dissipation and/or thermal limits of the switches. These next few plots examine the power dissipation of a switching circuit. We will use two different techniques to calculate the power dissipated during various cycles of the steady state switching waveform.

For the turn on and turn off transitions, we will make the reasonable assumption that the currents and voltage make linear transitions from their on states to off states, and vice-versa. This means we will calculate the total energy dissipated as .

LTspice will also perform an integration for us. We first set the time scale of the plot to the value we desire, slightly cutting off the edges to eliminate large spikes. We first move the cursor down to the x-axis until we see the yellow ruler icon and then right click to enter the axis limits as 19.0001 and 19.0499. Then move the cursor over the label of the quantity of interest and enter Ctrl-Left-Click to calculate both the average and the integral of the quantity over the horizontal axis limits. One can also use this technique to calculate the energy lost during the rising and falling transitions approximated above as a triangular waveform.

1. Plot the power dissipated in the IGBT (V(igbtc)-V(igbte))\*Ix(U1:C) over the entire waveform. Include the values of the power. What causes the change in the power from the initial startup to its steady state value?
2. Measure the energy dissipated during each on transition. First measure , then measure , and then measure . From these numbers calculate . Use the integral calculation of LTspice to calculate . How accurate is the triangular approximation?
3. Measure the energy dissipated during each off transition. First measure , then measure , and then measure . From these numbers calculate . Use the integral calculation of LTspice to calculate . How accurate is the triangular approximation?
4. Measure the energy dissipated in the IGBT during its on state
5. Add these to get the total energy dissipated in one cycle .
6. Calculate the average power during steady state .

## Buck Regulator with SiC NMOS transistor

### Setup

We have analyzed the behavior of a modern IGBT in our circuit. Now we will perform this analysis using a modern NMOS transistor. High band-gap semiconductors using silicon carbide were introduced into production around 2010. The high band-gap allows the semiconductor to hold off more voltage than does silicon. This allows MOSFETs to be used in circuits at these power levels. We will replace the IGBT we used in the previous example with a MOSFET and examine the performance differences between the two.

1. Delete component and labels
   1. Delete IGBT ikw75n65eh5
   2. Delete labels igbtC (one place), igbtB (two places), and igbtE (two places)
2. Add components and labels
   1. NMOS transistor
      1. Type P to place a component
      2. In top line, pull down menu and select custom directory (C:\files\ltSpiceFiles\sym)
      3. Select nmosC3M0015065D (650V, 96A), click OK
      4. Place the transistor in the circuit where the IGBT was and ensure it is properly connected.
   2. Place a DC voltage source with an output of Vambient = 25. Connect the negative terminal to ground.  
      Note that the LTspice model of this device calculates an internal junction temperature of the NMOS. LTspice treats it as a voltage, although the user should interpret the voltage as a temperature reading in which one volt equals one degree Centigrade. Vambient sets the ambient temperature in which the device operates. The LTspice model uses this value as a reference for the case temperature, Tc. One can then measure Tj to obtain the junction temperature.
   3. Add labels nmosD (one place), nmosG (two places), nmosS (two places), tAmbient (two places), and tJunction (one place)
3. Change some LTspice directives
   1. Change the voltage levels of the gate pulser to -4 V and 15 V to comply with the recommended values from the vendor.
   2. Change the transient analysis command to deselect the “Start external DC supply voltages at 0V:”. This eliminates “startup” from the transient command. We want the Vambient to start at 25 °C instead of 0 °C so that we get an accurate plot of the NMOS junction temperature.
4. Save the schematic as buckNmos10k.asc



Figure 9: NMOS buck regulator schematic

### Analysis

We have already analyzed the general behavior of the circuit. This is independent of the switching device we use. Here we will repeat the measurements that depend on the device, such as switching times, on resistances, and power dissipation.

1. On one plot sketch two cycles of the drain-to-source NMOS voltage (V(nmosd)-V(nmoss)) and current Ix(U1:D) under steady state conditions. Include the “on” and “off” values of the NMOS voltage and the value of the current during the cycle. The voltage drop across the NMOS in the “on” state can be written as a linear equation . From the data, calculate and . How do these numbers compare with those of the IGBT?
2. Measure the energy dissipated during each on transition. First measure , then measure , and then measure . From these numbers calculate . Use the integral calculation of LTspice to calculate . How accurate is the triangular approximation?
3. Measure the energy dissipated during each off transition. First measure , then measure , and then measure . From these numbers calculate . Use the integral calculation of LTspice to calculate . How accurate is the triangular approximation?
4. Measure the energy dissipated in the MOSFET during its on state
5. Add these to get the total energy dissipated in one cycle .
6. Calculate the average power during steady state .
7. How do these numbers compare with those of the IGBT?
8. Sketch the inductor current, MOSFET power dissipation, and junction temperature over the full , showing their peak values. What is the relationship between the three? What is the approximate time delay between when power is dissipated on the transistor and when the junction temperature reacts to it?

## Buck Regulator, Modified (Step Change in Operating Point)

We will now modify the circuit slightly to see how it responds to a step change in its operational point.

We do this by adding a second voltage source in series with V2 and setting the pulses of each so that:

* They each have different duty cycles
* They are set to zero volts while the other source is pulsing

1. Load buckNmos10k.asc: File->Open buckNmos10k.asc
2. Stretch (S) the components in the pulser subcircuit to allow the addition of another voltage source.
3. Delete (Del) the wire connecting V2- to the nmosS network, escape to exit
4. Add a voltage source by selecting V, place below V2, and connect the two in series
5. Wire (W) the new voltage source, V3, to be in series with V2, with one side connected to V2 and the other side to the nmosS network.
6. Verify the PULSE parameters of V2 are set as follows. We will only pulse this for 200 cycles.
   1. Vinitial = -4; Von = 15; Tdelay = 0; Trise = 10n; Tfall = 10n; Ton = 50u; Tperiod = 100u; Ncycles = 200
7. Set PULSE parameters of V3. We will hold this off for the first 20 ms while V2 is pulsing, then turn it on at a different duty cycle for another 20 ms. Since V2 will be at -5 V when it is not pulsing, including when it has stopped pulsing, V3 needs to pulse between 0 V and 19 V.
   1. Right click on V3
   2. Advanced
   3. Select PULSE function
   4. Vinitial = 0; Von = 19; Tdelay = 20m; Trise = 10n; Tfall = 10n; Ton = 60u; Tperiod = 100u; Ncycles = 200
   5. OK
8. Modify the simulation command to calculate for 40 ms
   1. Simulate->Edit Simulation Cmd
      1. Transient
      2. Stop time: 40m (this is 2NcyclesTperiod)
      3. Time to start saving data: 0
      4. Do not select external voltages to start at zero.
      5. OK
   2. This gives a simulation command of .tran 0 40m 0
9. Save file File->Save As buckNmosDual10k.asc



Figure 10: NMOS buck converter, two voltages

1. Run the simulation and examine the behavior, especially at t=20 ms, when the supply changes its operating point. Sketch the change in output voltage during the transition. Compare the dynamics of the circuit, such as overshoots and response times of the two transitions, 0 - 30 A and 30 – 36 A.
2. What are the relations between the voltage across the inductor and the current through it at 39 ms, when the second step has reached its steady state value? What is the average voltage across the inductor when it is in steady state?

## Snubber Circuits

As we have seen, the switching devices dissipate power in several ways. One source of power dissipation is the current flowing through the device. This is caused by fixed voltage drops in the devices as well as internal resistances. This dissipation is independent of the device switching frequency. The other source of power dissipation is energy dissipated when the device is transitioning between the on and off states. Since this dissipation occurs during every switch transition, this power is linearly dependent on the switching frequency. The more transitions per unit time, the more energy lost.

1. Load buckNmos10k.asc
2. Run the simulation for 20 ms. Use the LTspice tools to measure the junction temperature and total energy lost during that time.
   1. Record the peak junction temperature \_\_\_\_\_\_\_\_.
   2. Record the average junction temperature over the 20 ms \_\_\_\_\_\_\_.
   3. Record the total energy dissipated over the 20 ms \_\_\_\_\_\_\_\_.
3. Modify the circuit to have a 50% duty cycle with a switching frequency of .
   1. The PULSE command should have one tenth of the on time and the period and ten times the number of cycles: PULSE(-4 15 0 10n 10n 5u 10u 2000)
   2. Save the file as buckNmos100k.asc
   3. Run the simulation file for 20 ms
   4. Record the peak junction temperature \_\_\_\_\_\_\_\_.
   5. Record the average junction temperature over the 20 ms \_\_\_\_\_\_\_.
   6. Record the total energy dissipated over the 20 ms \_\_\_\_\_\_\_\_.
   7. Record the peak power dissipated during the on transition of the MOSFET at 19 ms \_\_\_\_\_\_\_
   8. Record the energy dissipated during the on transition \_\_\_\_\_\_\_\_
   9. Record the peak power dissipated during the off transition of the MOSFET at 19 ms \_\_\_\_\_\_\_
   10. Record the energy dissipated during the off transition \_\_\_\_\_\_\_\_

Now we add some snubbers.

1. First save the file as buckNmosNoSnub100k.asc in preparation for modifying it.
2. Cut the trace between the pin of the MOSFET and the diode cathode.
3. Build the three structures, two snubbers and a straight wire, shown on the right hand side of Figure 11. Use the VS-E5PH7506 diode, the same type of diode used in the regulator, in each snubber.
   1. For the capacitor snubber use a capacitor and a resistor.
   2. For the inductor snubber use a inductor and a resistor.
4. Terminate all these subcircuits with grounds.
5. Resave the file.



Figure 11: NMOS buck regulator with snubber circuits

Run the simulations

1. First run with the nmosS label on one side of the wire and vDiode on the other. Ensure the two snubbers are terminated with grounds. Verify the simulation results are the same as those of the previous simulation.
   1. What is the maximum voltage across the transistor in the steady state, At what part of the waveform is this maximum reached?
   2. What is the maximum drain current of the MOSFET in the steady state, At what part of the waveform is the maximum reached?
2. As we add more elements to the simulation, the simulation time can get very long and sometimes stalls out. For the snubber study we want to look at the effect of the snubber on the transitions. We can do these calculations on a steady state situation. LTspice has fewer difficulties calculating systems that are close to their steady state solutions. Therefore, we will start them close to their steady state solutions by using the initial conditions LTspice directive. (Note that our initial conditions will be close, but not exact, because there are “hidden” elements in the model for which we cannot specify the initial conditions.)
3. Simulate the capacitor snubber in parallel with the MOSFET.
   1. Save the file as buckNmosCSnub100k.asc
   2. Modify the transient instruction to take 4 ms of data: .tran 0 4m 0
   3. Modify the pulse instruction to execute 400 cycles
   4. Add the initial condition command specifying the voltage on the C1 capacitor, Vout, Praeg capacitor, vPraeg, junction temperature, tJunction, and filter inductor current, I(L1).  
      .ic V(Vout)=150.01 V(vCPraeg)=150.36 V(tJunction)=31.5 I(L1)=33.2
   5. Replace the ground on the left-hand side of the capacitor snubber with the nmosD label.
   6. Replace the ground on the right-hand side of the capacitor snubber with the nmosS label.
   7. Resave the file.
   8. Record the peak power dissipated during the on transition of the MOSFET at 3.9 ms \_\_\_\_\_\_\_
   9. Record the energy dissipated during the on transition \_\_\_\_\_\_\_\_
   10. Record the peak power dissipated during the off transition of the MOSFET at 3.9 ms \_\_\_\_\_\_\_
   11. Record the energy dissipated during the off transition \_\_\_\_\_\_\_\_
   12. Record the average power dissipated in the device over the last millisecond of the simulation .
   13. On which part of the switching cycle does this snubber remove power dissipation in the MOSFET?
   14. What is the maximum voltage across the transistor in the steady state, At what part of the waveform is this maximum reached?
   15. What is the maximum drain current of the MOSFET in the steady state, At what part of the waveform is the maximum reached?
   16. Sketch the snubber capacitor current and voltage during both the turn-on and turn-off parts of the switching cycle.
   17. What is the mechanism by which this parallel snubber reduces the power in the MOSFET?
4. Simulate the inductor snubber in series with the MOSFET.
   1. Save the file as buckNmosLSnub4ms100k.asc
   2. Replace the ground on the left-hand side of the inductor snubber with the nmosS label.
   3. Replace the ground on the right-hand side of the inductor snubber with the vDiode label.
   4. Terminate the capacitor snubber and blank wire with two grounds, each.
   5. Resave the file.
   6. Record the peak power dissipated during the on transition of the MOSFET at 3.9 ms \_\_\_\_\_\_\_
   7. Record the energy dissipated during the on transition \_\_\_\_\_\_\_\_
   8. Record the peak power dissipated during the off transition of the MOSFET at 3.9 ms \_\_\_\_\_\_\_
   9. Record the energy dissipated during the off transition \_\_\_\_\_\_\_\_
   10. Record the average power dissipated in the device over the last millisecond of the simulation .
   11. On which part of the switching cycle does this snubber remove power dissipation in the MOSFET?
   12. What is the maximum voltage across the transistor in the steady state, At what part of the waveform is this maximum reached?
   13. What is the maximum drain current of the MOSFET in the steady state, At what part of the waveform is the maximum reached?
   14. Sketch the snubber inductor current and voltage during both the turn-on and turn-off parts of the switching cycle.
   15. On which part of the switching cycle does this snubber remove power dissipation in the MOSFET?
   16. What is the mechanism by which this parallel snubber reduces the power in the MOSFET?
5. Simulate both snubbers:
   1. Save the file as buckNmosLCSnub4ms100k.asc
   2. Replace the ground on the left-hand side of the inductor snubber with the nmosS label.
   3. Replace the ground on the right-hand side of the inductor snubber with the vDiode label.
   4. Replace the ground on the left-hand side of the capacitor snubber with the nmosD label.
   5. Replace the ground on the right-hand side of the capacitor snubber with the nmosS label.
   6. Resave the file.
   7. Record the peak power dissipated during the on transition of the MOSFET at 3.9 ms \_\_\_\_\_\_\_
   8. Record the energy dissipated during the on transition \_\_\_\_\_\_\_\_
   9. Record the peak power dissipated during the off transition of the MOSFET at 3.9 ms \_\_\_\_\_\_\_
   10. Record the energy dissipated during the off transition \_\_\_\_\_\_\_\_
   11. Record the average power dissipated in the device over the last millisecond of the simulation .
   12. What is the maximum voltage across the transistor in the steady state, At what part of the waveform is this maximum reached?
   13. What is the maximum drain current of the MOSFET in the steady state, At what part of the waveform is the maximum reached?
   14. By what factor did the two snubber circuits reduce the power dissipation in the transistor?