# Instructions for LTspice Laboratory Exercise

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 Ω. The inductor in the LPF has a value of $500\ \mu H$ and the capacitor has a value of $1000\ \mu F$.
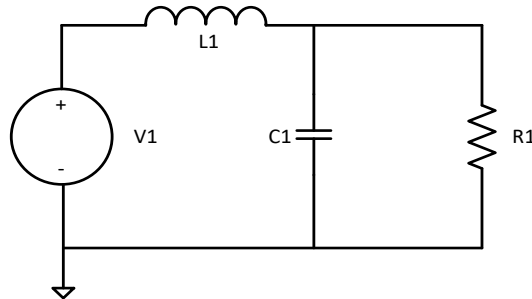
1. Open a new schematic: File->New Schematic (Ctrl-N)
2. Add components
   a. Voltage source: Edit->Component, F2, or AND gate icon from menu->
      i. Ensure top directory is ..\LTspiceXVII\lib\sym
      ii. Select voltage
      iii. Place and escape to exit
   b. Inductor: Edit->Inductor or select from menu, place, Ctrl-R to rotate, escape to exit
   c. Capacitor: Edit->Capacitor or select from menu, place, escape to exit
   d. Resistor: Edit->Resistor or select from menu, place, escape to exit
   e. Ground: Edit-> GND or select from menu, place, escape to exit
3. Connect components: Edit->Draw Wire, F3, or select Line icon from menu
   a. Click on GND, draw to V- and click
   b. Click on V+, draw in right angles to LHS of inductor, clicking on the corners and inductor
   c. Click on RHS of inductor, draw in right angles to top of resistor, clicking on corner of resistor
   d. Click on bottom of resistor, draw in right angles to bottom of line between GND and V-, clicking on corners and termination line
   e. Click on line from inductor to resistor and click on top of capacitor
   f. Click on line from resistor to ground and click on bottom of capacitor
   g. Escape to exit
4. Set the values of the components
   a. Inductor: Right click on the inductor, enter 500u in inductance box, OK
   b. Capacitor: Right click on the capacitor, enter 1000u in capacitance box, OK
   c. Resistor: Right click on the resistor, enter 5 in resistance box, OK
   d. 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

$$f = \frac{1}{2\pi} \frac{1}{\sqrt{L1 \cdot C1}} = \frac{1}{2\pi} \frac{10^6}{\sqrt{5 \cdot 10^5}} = 225 \, Hz$$

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
    a. Edit->Spice Analysis or Simulate->Edit Simulation Command ->AC Analysis
        i. Type of sweep: Decade
        ii. Number of points per decade: 1000
        iii. Start frequency: 10
        iv. Stop frequency: 10k
        v. OK
    b. Or Edit->Spice Analysis->.ac dec 1000 10 10k, OK
    c. Click on schematic to place command
2. Save schematic as lcFilterFreq.
3. Run simulation: Select runner icon, or Simulate->Run
    (A netlist is created in lcFilterFreq.net. This contains the SPICE commands
    * C:\files\ltSpiceFiles\examples\2017\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 Label Net command (F4), 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.)
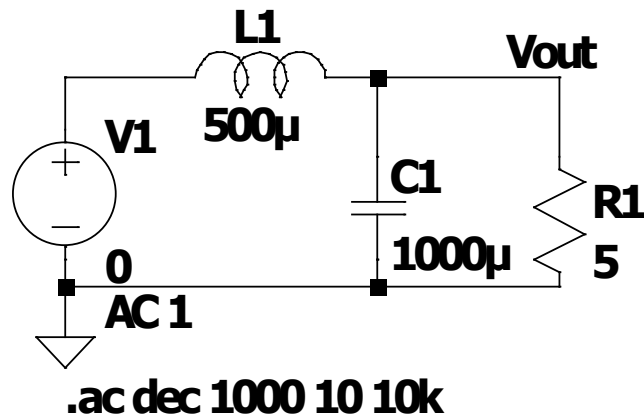
Figure 1: lcFilterFreq.asc

4. Plots:
   a. 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.
   b. 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.
   c. 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.)
   d. 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 using either Plot Settings->Add trace or Ctrl-A, 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(n00x)-V(n00y))*I(R1), OK, where n00x is the node on one side of R1 and n00y is the node on the other side.
   e. 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.
   f. 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.

5. 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 225 Hz (it is a little lower because of the resistor)

6. The phase of the response changes from $0°$ to $-180°$ 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

7. Record the frequencies and amplitudes at which the phases are $-45°$, $-90°$, and $-135°$, respectively

   -45° _____Hz   _____dB

   -90° _____Hz   _____dB

   -135° _____Hz   _____dB

This shows the maximum amplitude occurs at approximately -90° and that the amplitudes are down approximately 3dB 45 degrees on either side of 90 degrees.

8. 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 (Edit->Delete, F5, or scissors 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. Edit->Spice Analysis or Simulate->Edit Simulation Command -> Transient
   a. Stop time: 50m
   b. Time to start saving data: 0
   c. Select "Start external DC supply voltages at 0V"
   d. 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: Select runner icon, or Simulate->Run
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, $v(\infty)$, to the amplitude of the first peak, with respect to $v(\infty)$. 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 τ using the formula $e^{-t/\tau}$ to fit the data.

$$\frac{v(t+T) - v(\infty)}{v(t) - v(\infty)} = \frac{V_o e^{-\frac{t+T}{\tau}} \sin \omega(t+T)}{V_o e^{-\frac{t}{\tau}} \sin \omega(t)} = e^{-\frac{T}{\tau}}$$

Since the value of $\sin \omega t$ is the same at the two measurement times, solving for τ we get

$$\tau = -\frac{T}{\ln \frac{v(t+T)}{v(t)}}$$

Ratio: _____

Damping time τ: _____

L1
V1
500μ
+
−
0
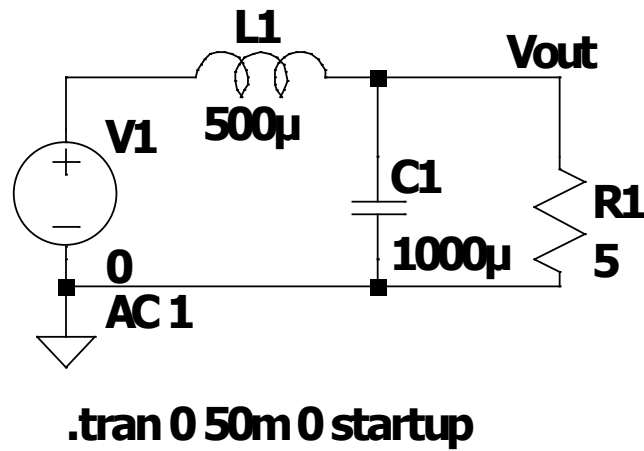AC 1
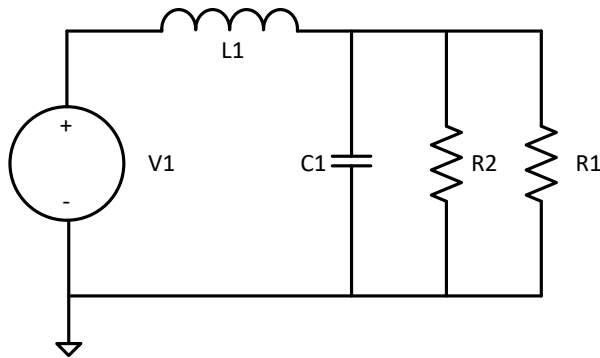C1
1000μ
Vout
R1
5

.tran 0 50m 0 startup

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.

L1
+
−
V1
C1
R2
R1

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. Depress the *s* key, use the *.op* command at the right hand side of the menu, or select the *SPICE Directive* from the *Edit* pull down menu. All of these ways opens a dialog box *Edit Text on the Schematic*. Ensure highlight of the *SPICE Directive* radio button. 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: Edit->Resistor or select from menu, place, escape to exit
3. Connect resistor to existing circuit using Edit->Draw Wire, F3, or select Line icon from menu. 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 $L - C$ circuit, the only quantity you can create with the units of impedance is $R = \sqrt{L_1/C_1}$. Although this may not be the value of the "best" damping resistor, it should be close to that value, so $R = \sqrt{L_1/C_1}$ is a good value to include in your parameterization.
4. 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 0.01 Ω to 10 Ω with four steps per decade.
5. Set up an AC analysis from 20 Hz to 2 kHz:
   a. Edit->Spice Analysis or Simulate->Edit Simulation Command ->AC Analysis
      i. Type of sweep: Decade
      ii. Number of points per decade: 1000
      iii. Start frequency: 20
      iv. Stop frequency: 2k
      v. OK

      b.   Or Edit->Spice Analysis->.ac dec 1000 20 2k, OK

      c.   Click on schematic to place command

      d.   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

6.   Save file as lcFilterDampFreq

7.   Run simulation: Select runner icon, or Simulate->Run

8.   Plot Settings->Select Steps allows you to select which of the parameterization values of R you wish to plot

9.   Replace the .step command with a command that will vary the parameter linearly from 0.5 Ω to 2 Ω in steps of 0.1 Ω. Right click on the .step command to bring up the .step Statement Editor:

      a.   Either use the prompts

           i.   Name of parameter to sweep: R

          ii.   Nature of sweep: Linear

         iii.   Start value: 0.5

         iv.   Stop value: 2.0

          v.   Increment: 0.1

      b.   Or enter .step param R 0.5 2.0 0.1 in the bottom box

      c.   OK

10.   Run simulation: Select runner icon, or Simulate->Run

11.   Choose the desired resistor value (when the output voltage is essentially critically damped). 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.

12.   Replace the Resistance value **{R}** with the desired value and record this value

_____

13.   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
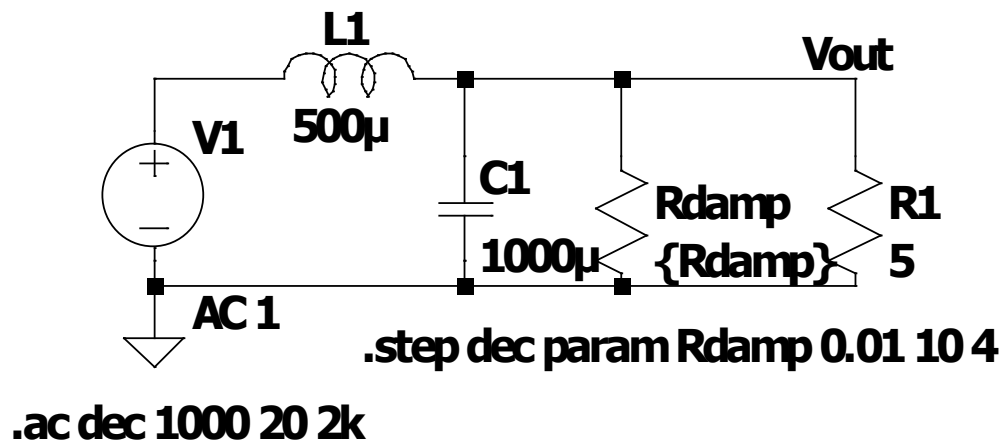
Figure 3: lcFilterDampFreq.asc

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
   a. DC value: 1
   b. AC Amplitude: 0
   c. OK
2. Right click on the .ac command to open the Edit Simulation Command box and
   a. Either
      i. Select Transient tab
      ii. Stop time: 10m
      iii. Time to start saving data: 0
      iv. Select Start external DC supply voltages at 0V
   b. Or in lower box enter
      i. .tran 0 10m 0 startup
   c. OK
3. Save file as lcFilterDampStep.asc
4. Run simulation: Select runner icon, or Simulate->Run
5. Sketch the circuit output voltage. Sketch and record the power dissipated in the resistor (see sections 4.c and 4.d in the discussion of Frequency Response - Plots above).
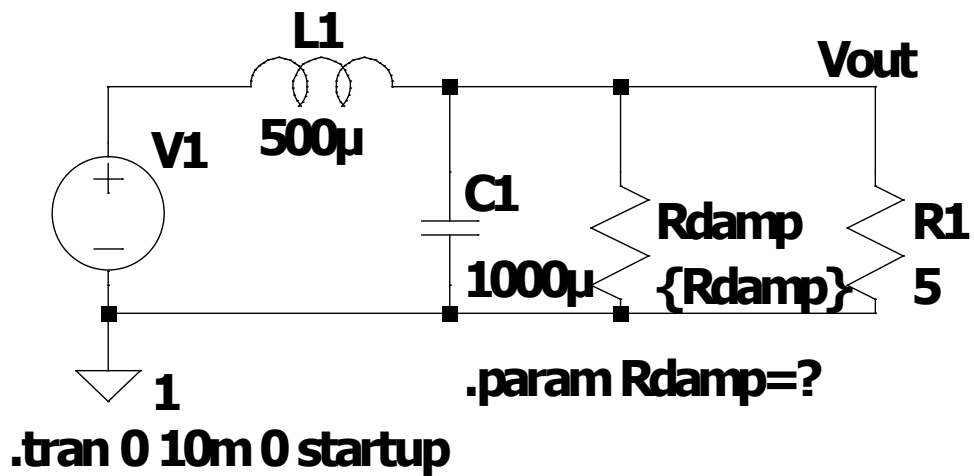
a. Output Voltage

b. R2 Dissipated Power

L1

Vout

V1 500μ

C1

Rdamp R1

1000μ {Rdamp} 5

.param Rdamp=?

1

.tran 0 10m 0 startup

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.

L1

V1

C1

R2

R1

C2
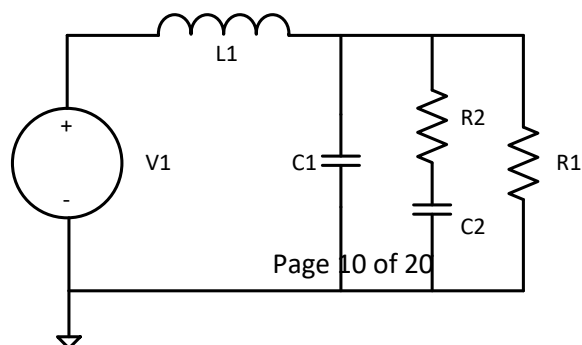
Parametric Analysis

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:
    a. Edit->Spice Analysis or Simulate->Edit Simulation Command ->AC Analysis
        i. Type of sweep: Decade
        ii. Number of points per decade: 1000
        iii. Start frequency: 20
        iv. Stop frequency: 2k
        v. OK
    b. Or Edit->Spice Analysis->.ac dec 1000 20 2k, OK
    c. Click on schematic to place command
3. Delete the .step statement: F5, select, escape
4. Modify the damping resistor (R2) 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 R2. F7 is Move, F8 is Drag. Drag is similar to move, but wires rubber band along with selected objects.
6. Delete the wire connecting R2 to ground: F5, select, escape
7. Add a capacitor: Edit->Capacitor or select from menu, place, escape to exit
8. Connect the capacitor to R2 and GND using Edit->Draw Wire (F3), escape to exit
9. Set C2 to be parameterized
    a. Right click on C and set Capacitance to {C}
    b. Add the SPICE command to tell LTspice the parameters of interest: Edit->SPICE Directive
        .step param C 3000u 7000u 1000u
        OK then place
        to step through five values of the capacitance.
10. Save the file as praegFreq.asc: File->Save As praegFreq.asc
11. Run simulation: Select runner icon, or Simulate->Run
12. Select and record an acceptable capacitor value for this load
    Hint: From class notes $C_2 = 5 * C_1$

_____ µF

AC Analysis

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

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

_____ Hz, _____dB

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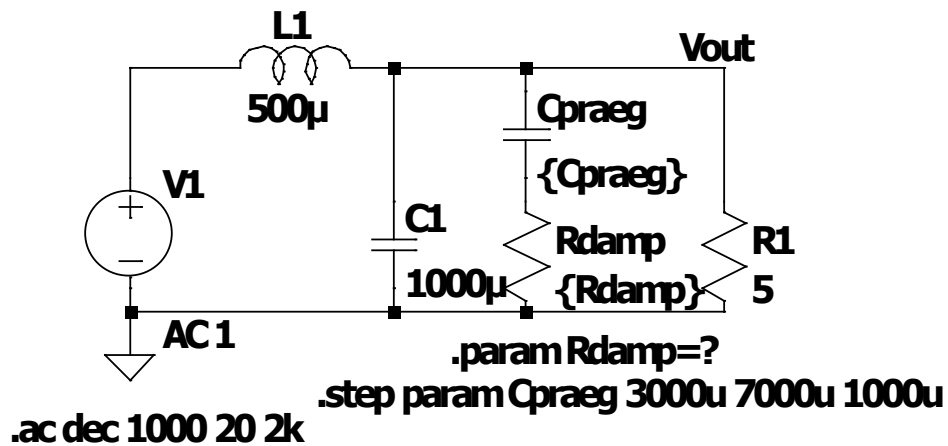

Figure 5: praegFreq.asc

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
   a. DC value: 1
   b. AC Amplitude: 0
   c. OK
2. Right click on the .ac command to open the Edit Simulation Command box and
   a. Either
      i. Stop time: 10m
      ii. Time to start saving data: 0
      iii. Select Start external DC supply voltages at 0V
   b. Or in lower box enter
      i. .tran 0 10m 0 startup
   c. OK
3. Save file as praegStep.asc
4. Run simulation: Select runner icon, or Simulate->Run

5. Select desired capacitance and replace the parameter C with that value. Delete the .step statement.
6. Sketch the output voltage versus time

Sketch the power dissipated in the damping resistor $(V(n002) - V(n001)) * I(R_2)$ over 50ms. Note that the steady state resistor power is zero
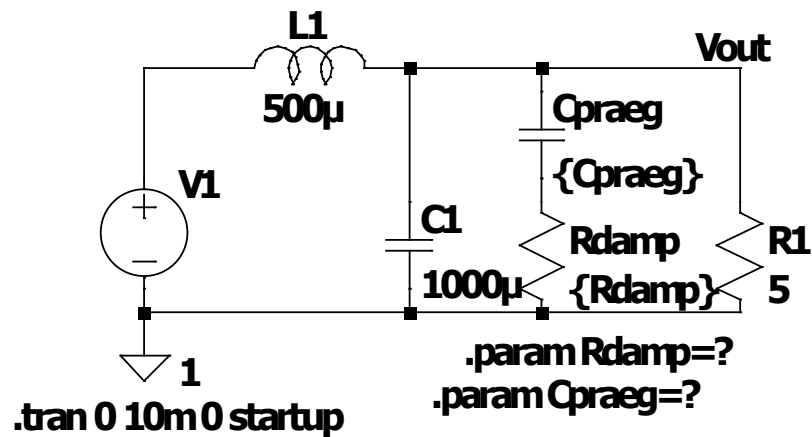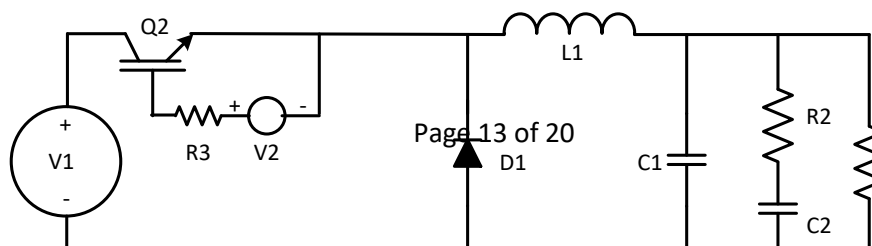

Figure 6: praegStep.asc

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.

Initial Response

1. Load praegFreq.asc: File->Open praegFreq.asc
2. Ensure that the parameters for R2 and C2 have been replaced by the desired components and delete (F5) any unneeded step statements
3. Drag (F8 and select) 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:
    a. IGBT:
        i. Edit->Component, F2, or AND gate icon from menu
        ii. In top line, pull down menu and select custom directory (ltSpiceFiles\sym)
        iii. Select irgp4063dpbf, click OK
        iv. Rotate the device (Ctrl-R) until the gate faces down
        v. Place and escape to exit
    b. Voltage source:
        i. Edit->Component, F2, or AND gate icon from menu
        ii. Ensure top directory is ..\LTspiceXVII\lib\sym
        iii. Select voltage, OK, rotate (Ctrl-R) until positive terminal faces left
        iv. Place and escape to exit
    c. Diode:
        i. Edit->Component, F2, or AND gate icon from menu
        ii. Ensure top directory is ..\LTspiceXVII\lib\sym
        iii. Select diode, OK, rotate (Ctrl-R) until the cathode faces up
        iv. Place and escape to exit
    d. Resistor: Edit->Resistor or select from menu, rotate (Ctrl-R) until it is horizontal, place, escape to exit
5. Modify schematic
    a. Delete (F5) trace between V1 and L1, escape to exit
    b. Move (F7) components to look like figure.
    c. Draw Wire (F3) to connect the devices as in the figure.
6. Set the remaining component values
    a. Set V1 to 100 VDC: Right click on V1, DC value 100, AC Amplitude 0, OK
    b. Set R3 to 10 Ω: Right click on R3, Resistance 10, OK
    c. Set D1 to carry at least 20 A with a breakdown voltage greater than 250 VDC (conservative choices)
        i. Right click on D1
        ii. Pick New Diode
        iii. Select RF1501NS3S

iv. OK
d. Set V2 to be the gate drive, a pulse train to give a pulse-width modulated control of the IGBT. A gate value of -5 V keeps the IGBT off and a gate value of 20 V turns the IGBT on. We will start the pulse train immediately (Tdelay = 0), set the pulse period for $100\ \mu s$, set its duty cycle to 25% (Ton = Tperiod/4), and set some reasonable rise and fall times. Based on our filter response we expect the filter to settle out within a few ms, we set the number of cycles to 500 for a 50 ms excitation time. We will see the command

   PULSE(-5 20 0 2u 2u 25u 100u 500) by V2

   i. Right click on V2
   ii. Advanced
   iii. Select PULSE function
   iv. Vinitial = -5; Von = 20; Tdelay = 0; Trise = 2u; Tfall = 2u; Ton = 25u; Tperiod = 100u; Ncycles = 500
   v. OK

7. Set up a transient simulation:
   a. Simulate->Edit Simulation Cmd
      i. Transient
      ii. Stop time: 50m (this is Ncycles·Tperiod)
      iii. Time to start saving data: 0
      iv. OK
   b. This gives a simulation command of .tran 0 50m 0
8. Save file File->Save As buckInitial.asc
9. Run simulation: Select runner icon, or Simulate->Run
10. Sketch $V(R_1)$, Io $I(R_1)$, on one plot

11. Sketch $I(C_1)$, the current in the main Praeg capacitor C1 over 50ms and then over two cycles under steady-state conditions

12. On one plot sketch two cycles of the filter inductor L1 voltage (V(n002)-V(n003)), and current (IL1) under steady state conditions

13. On one plot sketch two cycles of the free-wheeling diode VD1, and ID1 under steady state conditions

14. On one plot sketch two cycles of the drain-to-source IGBT voltage (V(n001)-V(n002)) and current (-I(V1)) under steady state conditions

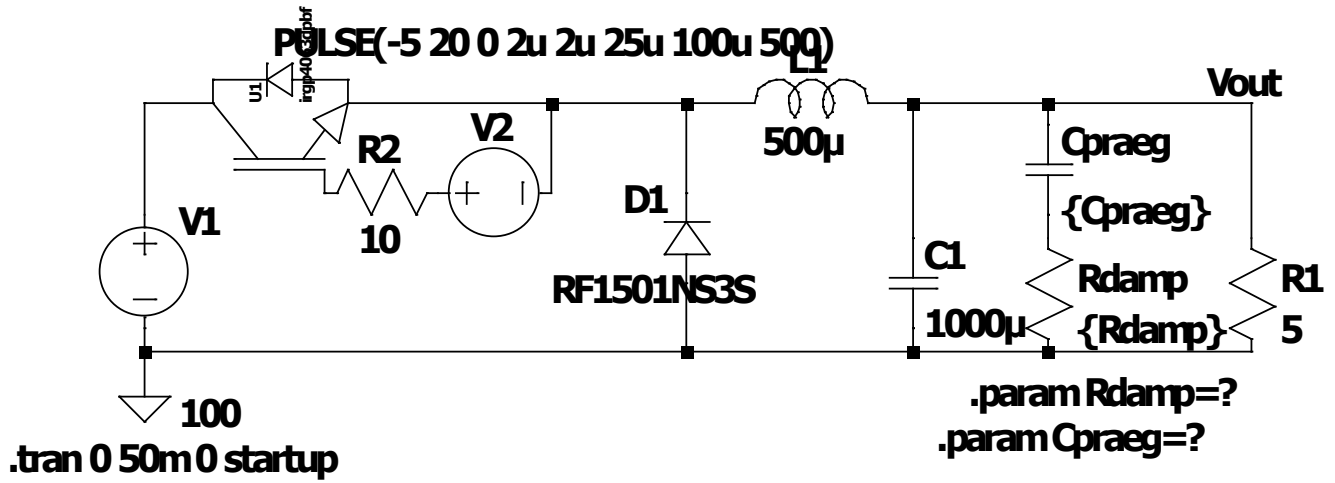15. On one plot sketch two cycles of the input voltage (V1) and current (-I(V1)) under steady state conditions

PULSE(-5 20 0 2u 2u 25u 100u 500)

L1

Vout

U1
irgib4063dpbf

R2

V2

500µ

V1

10

D1

RF1501NS3S

Cpraeg

{Cpraeg}

C1

Rdamp

R1

1000µ

{Rdamp}

5

100

.param Rdamp=?

.param Cpraeg=?

.tran 0 50m 0 startup

Figure 7: buckInitial.asc

U1
irgib4063dp

igbtE

L1

Vout

500µ

V1

igbtG

D1

RF1501NS3S

C1

Cpraeg

{Cpraeg}

Rdamp

R1

1000µ

{Rdamp}

5

100

.param Rdamp=?

.param Cpraeg=?

.tran 0 50m 0 startup

R2

LSE(-5 20 0 2u 2u 25u 100u 500)
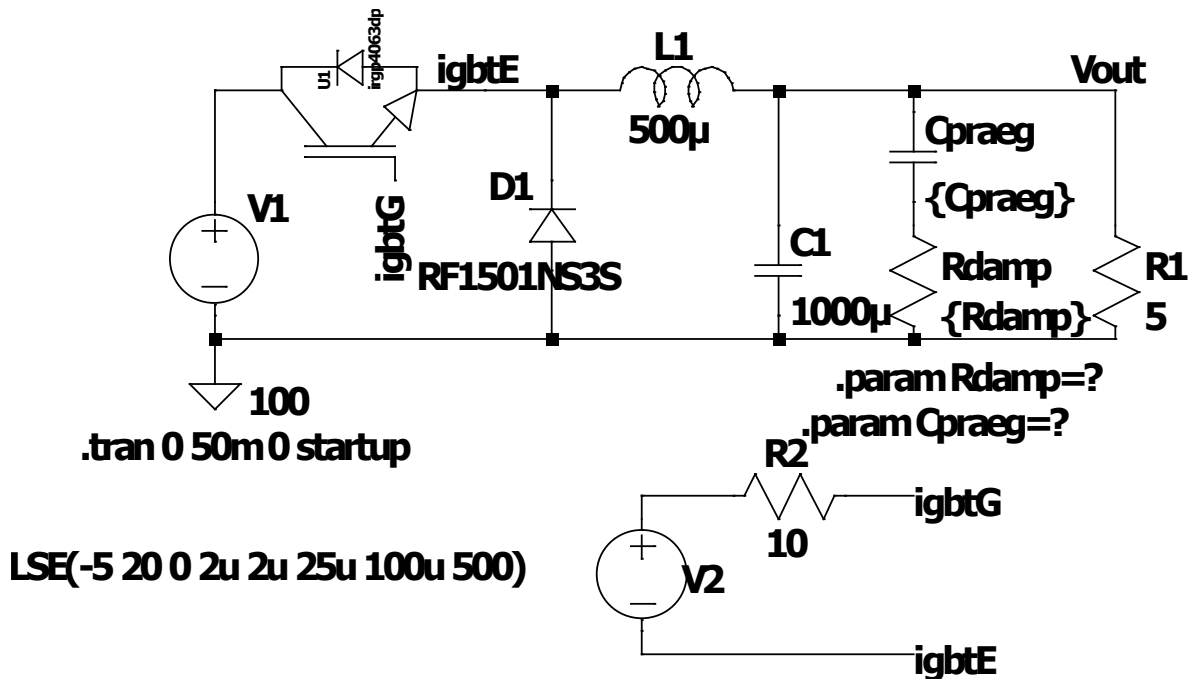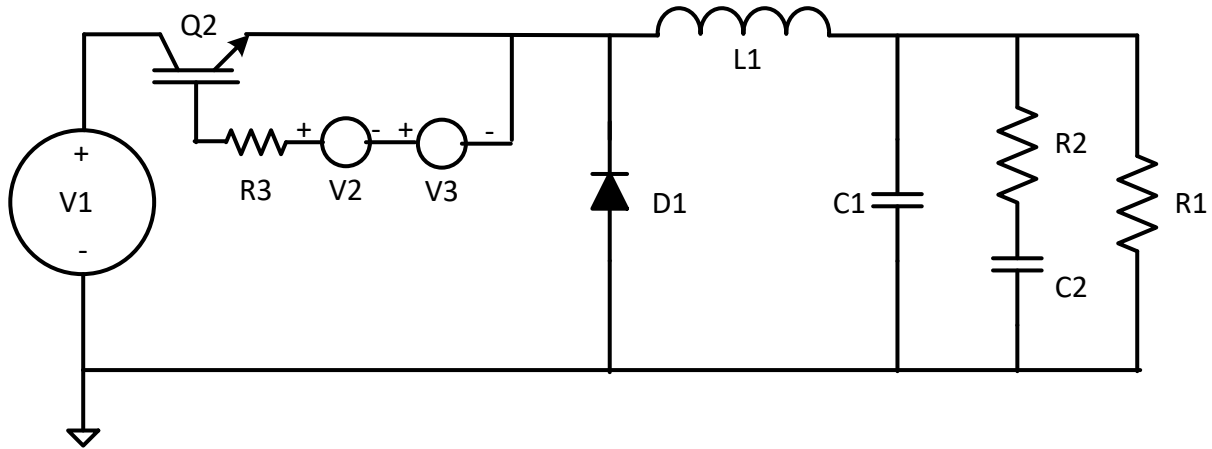
igbtG

10

V2

igbtE

Figure 8: buckNet.asc

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

1. Load buckInitial.asc: File->Open buckInitial.asc
2. Drag (F8, select, and drag) the diode and passive components, if necessary, to get extra space. Select to place and escape to exit.
3. Delete (F5) the wire connecting V2- to the IGBT emitter, escape to exit
4. Add a voltage source:
    a. Edit->Component, F2, or AND gate icon from menu
    b. Ensure top directory is ..\LTspiceXVII\lib\sym
    c. Select voltage, OK, rotate (Ctrl-R) until positive terminal faces left
    d. Place and escape to exit
5. Wire the new voltage source, V3, to be in series with V2.
    a. Draw Wire (F3)
    b. Place at each destination
    c. Escape to exit
6. Set PULSE parameters of V2. We will only pulse this for 200 cycles.
    a. Vinitial = -5; Von = 20; Tdelay = 0; Trise = 2u; Tfall = 2u; Ton = 25u; 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, V2 needs to pulse between 0 V and 25 V.
    a. Right click on V3
    b. Advanced
    c. Select PULSE function

      d.   Vinitial = 0; Von = 25; Tdelay = 20m; Trise = 2u; Tfall = 2u; Ton = 30u; Tperiod = 100u; Ncycles = 200

      e.   OK

8. Modify the simulation command to calculate for 40 ms
      a.   Simulate->Edit Simulation Cmd
          i.   Transient
          ii.   Stop time: 40m (this is 2·Ncycles·Tperiod)
          iii.   Time to start saving data: 0
          iv.   OK
      b.   This gives a simulation command of .tran 0 40m 0

9. Save file File->Save As buckSteadyState.asc
10. Run simulation: Select runner icon, or Simulate->Run
11. Examine the behavior, especially at t=20 ms, when the supply changes its operating point. Sketch the change in output voltage during the transition.

U1
irgit4063dpbf
igbtE
L1
500μ
Vout
V1
igbtG
D1
Cpraeg
{Cpraeg}
RF1501NS3S
C1
1000μ
Rdamp
{Rdamp}
R1
5
100
.param Rdamp=?
.param Cpraeg=?
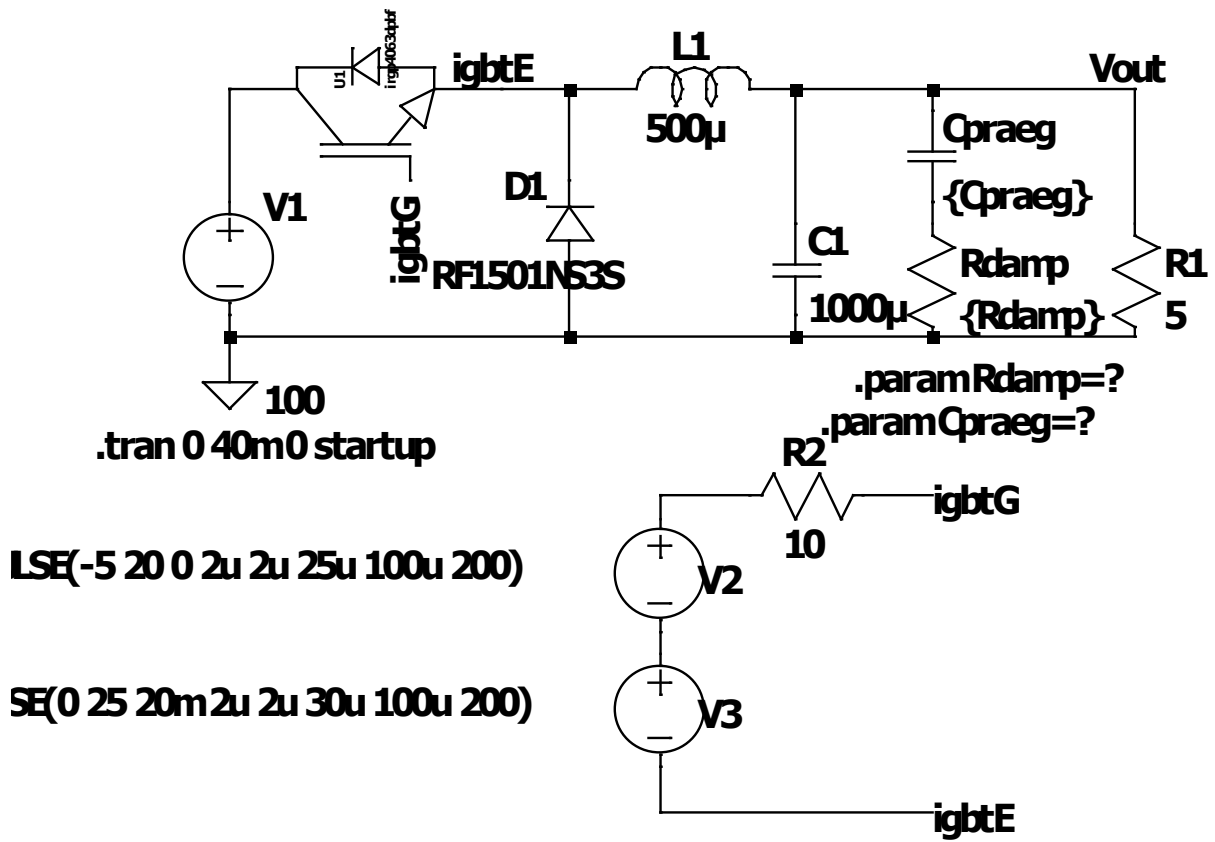.tran 0 40m 0 startup
R2
10
V2
igbtG
LSE(-5 20 0 2u 2u 25u 100u 200)
V3
SE(0 25 20m 2u 2u 30u 100u 200)
igbtE

Figure 9: buckSteadyState.asc