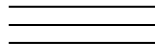


A Step-by-Step Instruction Set for Simulating Power Electronics using PSpice*(Release 9.1) with Schematics

* PSpice is a registered trademark of the Cadence Design Systems
(<http://www.Cadence.com>).



Ned Mohan

Modified by William P. Robbins

For the NSF/ONR-sponsored Workshop on Teaching of Power Electronics and Electric
Drives – Minneapolis, MN July 10, 2003

Table of Contents

	Page
1. Introduction	4
1.1 About PSpice	
1.2 Objectives of this Instruction Set	
1.3 Installing PSpice from Workshop 2003 CD-ROM	
1.4 Configuring Pspice to Run Power Electronics Files	
1.5 Simulation as a Three-Step Process	
2. Getting Started Using an Example	6
2.1 Open a Schematic File	
2.2 Simulate	
2.3 Plotting Results using Probe	
3. Getting to know PSpice by an Example	8
3.1 Schematic on the screen	
3.2 Setting up the Simulation Profile	
3.3 Probe Setup for Plotting	
4. Drawing a Schematic from Scratch	11
4.1 Linking the symbol libraries to the schematic	
4.2 Bringing parts to the schematic	
4.3 Setting up Simulation Profile	
4.4 Running the Simulation	
5. PSpice-Supplied Libraries and Components	14
6. Library PE_Lib1 and Model Library PE_lib: (supplied on the diskettes)	16
6.1 Diodes, MOSFETS, and Switches	
6.2 PowerPole Average Models	
6.3 PWM Controllers	
7. Fourier Analysis	18

8. Parametric Analysis	17
9. Performance Analysis in Probe	18
10. Analysis of Waveforms using built-in Macros within Probe	19
11. Frequency Domain Analysis	19
12. Avoiding and Overcoming Convergence; Other Useful Hints	20
Appendix - Description of PSpice Tutorial Circuits	21

1. Introduction

1.1 About PSpice

SPICE (Simulation Program with Integrated Circuit Emphasis) was developed at the University of California Berkeley. PSpice is a commercial version, a registered trademark of Cadence Design Systems (<http://www.cadence.com>).

1.2 Objectives of this Instruction Set

This instruction set describes a very small subset of PSpice Release 9.1 capabilities, specifically for simulating and designing power electronics systems:

- Creating new circuit schematics
- Transient analysis, parametric study, and Fourier analysis
- Frequency-domain analysis for designing feedback controllers
- Plotting of results using Probe; Performance Analysis
- Hints to avoid and overcome convergence problems

1.3 Installing PSpice

The Workshop 2003 CD contains the files needed for installing the student edition of Pspice 9.1. They are located in a subfolder entitled Install Pspice which in turn is located in the Pspice folder. Locate the file entitled Setup in the Install Pspice folder and activate it. When the Select Schematic Editor opens and requests a choice, choose Schematics. The Capture schematic editor will also be installed, but we recommend using Schematics only. Use default values when other windows request choices during the installation process.

1.4 Configuring Pspice to run Power Electronic Files

1.4.1 Automated Configuration

If you do not have any libraries which you have developed and want to remain configured in Pspice, you can use the ConfigurePspicePwrE folder located in the Pspice folder on the Workshop CD. In this folder,

locate the Setup.exe file and activate it. This will install the Pspice tutorial circuits, the power electronic component libraries, and configure them automatically by overwriting the existing Pspice configuration file (INI file) located in the Windows folder.

1.4.1 Manual Copying of Power Electronics Files.

Copy Pspice Tutorial Circuits folder (located inside the PSpice folder) on CD into the OrCAD_Demo folder on Drive C. C:\Program files\OrCAD_Demo will be the path name if the default values were used in the installation process. Open the folder Move to PSpice Userlib located in Pspice folder on CD. Select contents of Move to PSpice Userlib folder and copy these contents to the Userlib folder located in the Pspice folder on Drive C. The path name should be C:\Program files\OrCAD_Demo\PsPice\Userlib if the default values were used in the installation process.

1.4.2 Manual Configuration PSpice to Use Power Electronics Examples

The files moved into the Userlib folder must be configured as PSpice libraries in order for the circuit examples in the PSpice Tutorial Circuits folder to work.

Launch Schematics and open a new page using New in the File menu.

To configure the symbol file, PE_lib1.slb, begin by opening the Options menu and selecting Editor Configuration. When Editor Configuration window opens, select Library Settings button. When the Library Settings window opens, select the Browse button. In the browser, open the Userlib folder (path name is C:\Programfiles\OrCAD_Demo\PsPice\Userlib) and open PE_lib1. The Browse window will close. In Library Settings window, the path name to PE_lib1 will appear in the Library Name bar. Select Add* to add PE_lib1 to the list of configured libraries. Then click OK to close Library Setting window. Finally click OK in the Editor Configuration window to close it.

The student version only allows 10 libraries to be associated with the Schematics at any point. Therefore to add PE_LIB1, you may need to delete one library. We will not be using the 'sourcetm' library so you can safely remove it.

To configure the model library, PE_lib.lib, (which contains diode MUR2020 and MOSFET IRF640), activate the Options pull-down menu and choose the Library and Include Files button. In Library and Include Files window, select the Browse button and in the Browse window, navigate to Userlib folder and select PE_lib.lib and open it. Be sure to select All Files in the Files of type selection slot. The Browse window closes and returns control to Library and Include Files window. In Library and Include Files window, select Add Library and click OK to exit window.

1.5 Simulation as a Three-Step Process

1. Open a circuit file under Schematics
2. Simulate
3. Plot results using Probe.

2. Getting Started Using an Example

(Note: Selecting and clicking refer to clicking the left mouse button once.)

2.1 Open a Schematic

- To open the schematic editor, click on the following: Start - Programs – PSpice Student – Schematics.
- Under the 'File' menu, click on 'Open' and select 'Schematics' for file types. Navigate or type in the following filename with the full path name, for example, C:\Program files\OrCAD_Demo\Pspice_Tutorial_Circuits and click on 'Open' button.

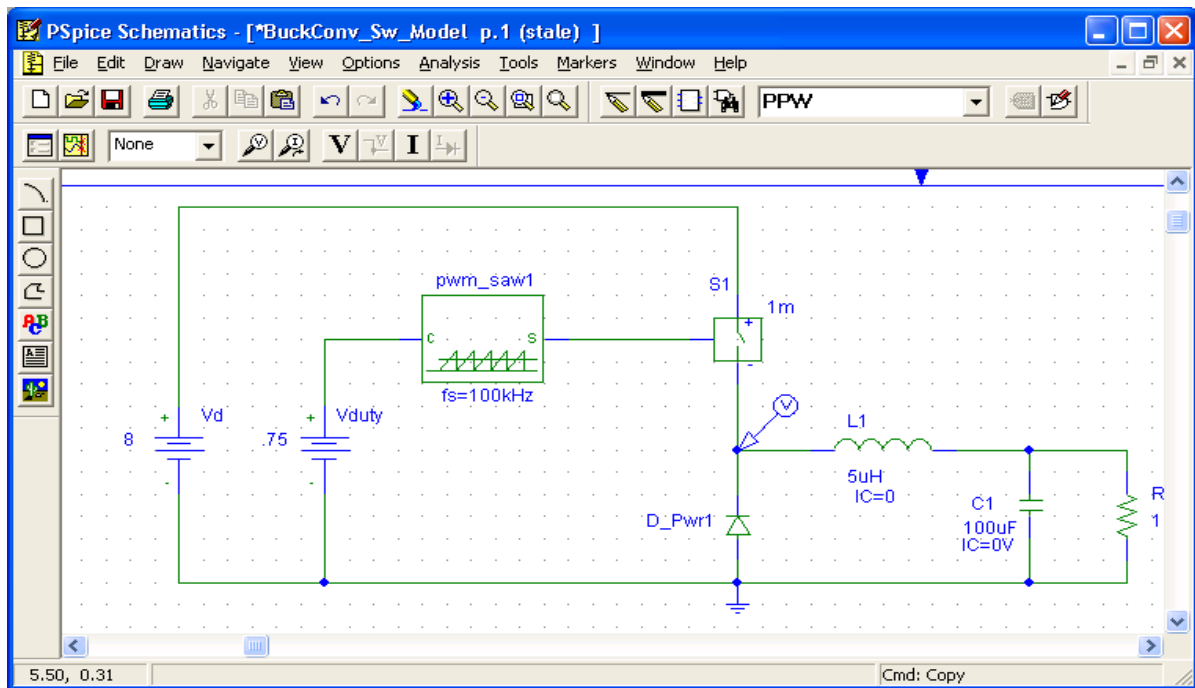



Fig. 1. Schematic file “Buck.sch”

2.2 Simulate

- Get familiar with the icons on the upper toolbars and the right-side toolbar by placing the cursor on each icon and waiting a few seconds for the explanation.
- Click on the run icon  to simulate (or under the 'Analysis' menu, click on 'Simulate').
- The PSpice simulation status window shows up which indicates the simulation status.
- This simulation is set up to run Probe automatically to plot the voltage and current, selected by the markers in the schematic, at the end of the simulation.

2.3 Plotting Results using Probe

- Look at the plotted results. Probe has created this plot using the raw plot points in the file Buck.dat created by the transient simulation.
- Exit probe by clicking on two 'X' buttons on the upper-most right hand corner.
- Under the 'Analysis' menu, select 'Examine Netlist'. This .net file is a listing of how PSpice has interpreted the buck converter drawn on the schematic. This

information is stored in a file called BuckConv_Sw_Model.net. Without Schematics, the user would have been required to create this file using a text editor. Close this file.

- Under the 'Analysis' menu, select 'Examine Output'. This is the output file BuckConv_Sw_Model.out. It will contain the error messages, if any, at the execution time. It will also contain the Fourier Analysis, if enabled.

3. Getting to know PSpice by an Example

3.1 Schematic on the screen

Now, we will explore various components, which make up the buck-converter schematic on our screen. Note that PSpice is case insensitive, that is, uppercase and lowercase letters mean the same. We have the option of using the following suffix letters to indicate various power-of-ten:

Suffix	value
f	10^{-15}
p	10^{-12}
n	10^{-9}
u	10^{-6}
m	10^{-3}
k	10^3
meg	10^6
g	10^9
t	10^{12}

- Each circuit should have one node designated as the ground node, which is connected to the 'AGND' symbol.
- Select and double click on the DC voltage source 'Vd'. A dialog box opens up. Uncheck "Include Non-changeable Attributes" and "Include System-defined Attributes". The dialog box then displays only the properties we would normally want to change. For the dc voltage source, we have only one

property to modify – 'DC'. Click on the entry 'DC' in the list box, and enter the value in the edit field above. Click 'Save Attr' to register the change. If you want to display the value on the schematic page, click 'Change Display', and select the required option under 'What to Display'.

- Double click on the inductor symbol. The inductor has a value of '5uH' where 'u' stands for 'micro' and 'H', the unit of inductance, is ignored by the program. The initial inductor current is A given in the 'IC' field. The inductor current direction is defined from pin1 to pin2.
- Similarly, double click on the capacitance (at the corner of one of its plates) and verify that it also can be assigned an initial condition. The capacitor voltage polarity is defined as the voltage at pin1 with respect to pin2.
- The diode in this schematic is a component in the symbol library PE_Lib1 supplied with your diskettes (with a part name 'D_pwr'). It uses a device model 'd' built-in within PSpice. Select the diode by double clicking near the cathode plate of the symbol and observe that its only parameter that can be changed is its resistance 'rs', which at present is assigned a value of $1\text{m}\Omega$.
- The switch is a voltage-controlled switch which is a part 'v-switch' in PE_Lib1. This switch symbol uses the PSpice model of a voltage-controlled switch designated by 'vswitch' which has a very low resistance R_{ON} in its on-state, when the control voltage is greater than V_{ON} . When the control voltage is less than V_{OFF} , the switch is represented by a very high resistance R_{OFF} . The default values are as follows: $R_{ON}=1\text{ ohm}$, $R_{OFF}=1\text{E}+6\text{ ohm}$, $V_{ON}=1\text{ V}$, and $V_{OFF}=0\text{ V}$. Note that the current through this switch can flow in either direction when the switch is on. Double click on this switch. Only its on-state resistance 'ron' can be changed, which, at present, is assigned a value of $1\text{m}\Omega$.
- In the PWM controller, the control voltage is compared with a high frequency sawtooth waveform, which establishes the switching frequency f_s which in this circuit is 100 kHz. The symbol PWM_SAW comes from the PE_Lib1 library. It has two terminals: *c*, where the control voltage is applied and *s*, where the output is provided to control the switch.

- Double-click on the symbol of the PWM controller to see the sub-schematic which makes it up. The control voltage at the c terminal is subtracted by a sawtooth waveform generated by a voltage source. Double click on the voltage source symbol and notice that in this ‘VPULSE’ source, the initial voltage level $V1$ is $0V$. It rises to $V2$ of $1V$ with a delay time TD of $0s$, a rise time TR of nearly $1/f_s$, and a fall time TF and a pulse width PW , both nearly *zero*. The period PER is $1/f_s$. Ideally, the rise time should be $1/f_s$, and the fall time and the pulse-width should be zero. In order to avoid convergence problems in PSpice, the fall time and the pulse width are given a small but finite value of 0.01 us . Notice that $\{ \}$ and $@$ are needed in this hierarchical arrangement, where the value of the frequency f_s is passed on from the buck schematic where it can be changed. The ABM block takes in two inputs, which are processed as per the EXP1 attribute. Notice how the two inputs are referenced in ‘EXP1’.
- How does this schematic in BuckConv_Sw_Model.sch link to the symbol library file PE_Lib1? This occurs because of the configuration procedure described in section 1.6.
- In the schematic, the Markers are placed on nodes for plotting their voltages and on component pins for currents going into those pins. Under the 'PSpice' menu, select ‘Markers’ and see that we have a choice of markers: voltage level (with respect to ground), differential voltage across two nodes, current into pin (or terminal) of a component, as well as advanced markers (such as voltage in dB) that become available if appropriate to the simulation.
- Note that all the components are connected by wires.

3.2 Setting up the simulation

Under the 'Analysis' menu click 'Setup'. In the dialog box that appears, make sure that the checkbox near ‘Transient’ pushbutton is checked. Click on ‘Transient’ to edit the settings for transient analysis. The most important

parameters for transient analysis are *Final Time*, the time till which the simulation is to be run, and *Step Ceiling*, the maximum time step the simulation can use. Also, make sure that *Skip initial transient solution* is checked. This prevents the program from calculating its own initial conditions based on bias-point calculations; rather the user-supplied initial conditions, if any, are used.


To setup probe, from 'Analysis' menu, click 'Probe Setup'. Notice that 'Automatically run Probe after simulation' is selected under 'At Probe Startup'. Select the 'Data Collection' tab where 'All' is selected. There is also a button for 'At markers only'. This option is important; it should be selected appropriately in a large circuit where collecting data at all nodes for a long transient simulation may result in an excessive simulation time and in a very large data file.

4. Drawing a Schematic from Scratch

Now, we will draw the same buck-converter schematic from scratch.

- Close the current schematic by clicking on the 'x' button the upper-right corner, discarding any changes.
- Go to the 'File' menu and click on 'New'. A new schematic window opens.

4.1 Linking the needed symbol libraries to this schematic


Before we bring in parts, make sure that we have the appropriate symbol libraries available for this schematic. Normally, a new schematic will have all the default PSpice libraries loaded. However, we will have to make sure that the symbol library supplied with the course is loaded properly. To check if it is so, click on 'Draw' menu, and click 'Place Part', (alternatively, use the toolbar icon ). In the Place Parts dialog box, click 'Libraries'. Make sure that 'PE_Lib1.slb' is included in the list of libraries. If not, use the procedure given in Sec. 1.6 to add the library. Click 'Cancel' to return to the Place Parts dialog box.

4.2 Bringing parts to the schematic

Click on the place part icon and the dialog box will open up. In the parts list, select 'VDC' (typing the name in the text box above the list helps in quickly searching for the part). Click 'Place & Close'. The dialog box closes, and you should be able to get the symbol of 'VDC' attached to the mouse. Left click the mouse to place the part at the desired location. While moving the part, press 'Ctrl-R' to rotate the part. Once you have placed the part, click 'Esc' to end the operation, or keep placing copies of the part by left-clicking the mouse on other locations on the schematic.

Do the same with the rest of the components to build the schematic in Fig. 1. Notice that the ground to be used is 'AGND'.


To manipulate existing components, select them by left-clicking the mouse. Drag the component to a new location, press Ctrl-R to rotate, and Ctrl-F to flip. To delete the component, press 'Del'.

To connect the components by wire, from 'Draw' menu, click 'Wire' (icon  from the toolbar). Make sure that the circuit is wired correctly.

4.3 Setting up Simulation

- Follow the procedure detailed in Sec.3.2.

4.4 Running the Simulation

- Save the Schematic: Under the 'File' menu, select 'Save'
- Start the Analysis: Start the simulation by clicking on the run icon .
- Exploring the Probe: Within Probe, explore many possibilities by pulling down various menus. For example in a plot, one or more waveforms can be scaled, waveforms can be labeled, certain portion of the plot can be zoomed in, limits on the axes can be manually specified, two or more waveforms can

be combined (for example, multiplied point-by-point) and plotted. More than one plot can be arranged one below the other, another Y-axis can be added.

- Printing Probe Waveforms: With waveforms on the screens select 'Page Setup' from the 'File' menu. Select all margins as 0.5 in, choose 'Portrait' and '3 plots/page', and cursor information on bottom. Click on 'OK'. Under 'File' menu choose 'Print'. Click on 'OK'.
- Taking the waveforms to a word processor: Under the 'Window' menu, choose 'Copy to clipboard'. Make sure to choose 'Make window and plot backgrounds transparent'. In the 'Foreground' box, check 'Change all colors to black' (unless you have a color printer). Now these waveforms can be pasted into the word processing program.
- Printing the Schematic: Switch to Capture so that your schematic is on the screen. By choosing 'Print' under the 'File' menu will print the entire page which will make the schematic very small. The suggested procedure is as follows: Draw a box around the part of the schematic you want to print. This will select it. Under 'Edit' Menu, click on 'Copy'. Paste it in a word processing program. In Microsoft Word, right click on the pasted schematic and format picture where its size can be adjusted and under 'Picture' tab, choose 'Black and White' in the color box.

5. PSpice-Supplied Libraries and Components

To draw a schematic, we need components which are stored in symbol libraries. All such libraries except for PE_Lib1 are part of PSpice loaded from the CD-ROM/downloaded file. This section contains the list of components from the PSpice libraries that are most likely to be used in power electronics simulations.

- Passive Components from the Analog library

Resistor R, Inductor L, Capacitor C, Coupled Inductors, XFRM_LINEAR

- Independent Voltage and Current Sources from the Source Library
Vdc, Idc, Vpulse, Ipulse, Isin and Vsin. For piece-wise-linear definition of voltage and current as a function of time, use VPWL and IPWL. Note that Vac and Iac are for sweeping frequencies in ac analysis (used in frequency-domain analysis).
- Dependent Voltage and Current Sources from the Analog Library
Voltage-controlled voltage source E, Current-controlled current source F, Voltage-controlled current source G, and Current-controlled voltage source H.
- Ground Symbol from the right-side Toolbar
The ground symbol to be used for simulations has to be AGND or GND_ANALOG or GND_EARTH.
- Avoiding Wires and using Bubble
Select 'BUBBLE' from the parts list (The component is part of library 'Port.slb'. Make sure that the library is linked to the schematic before using the part). By double clicking on the symbol, it can be renamed. Bubbles with the same name connect the nodes to which they are attached.
- Interface ports from library 'Port.slb'
Interface ports are used to connect nodes across multiple pages and also for hierarchical connections.
- Analog Controller Modeling using ABM Elements
 - Get an ABM element from the ABM library. Make sure that the ABM library is included. The output of this block is a voltage (with respect to ground) equal to the expression in this block. This expression can be changed by double clicking on the expression.

- Get an ABM1 element from the ABM library. The output of this block on the right side is a voltage (with respect to ground) equal to the expression in this block which relates the output to the input voltage $v(\%in)$. This expression can be changed by double clicking on the expression.
 - Get an ABM2 element from the ABM library. Similar to the ABM1 element, this element relates output to the two input voltages $v(\%in1)$ and $v(\%in2)$. Enter the entire expression in EXP1 and delete EXP2 entirely.
 - Get an ABM3 element from the ABM library. Similar to the ABM2 element, this element relates output to the three input voltages $v(\%in1)$, $v(\%in2)$ and $v(\%in3)$. Double click on the part to edit attributes. Delete EXP2 and EXP3 entirely and enter the expression in EXP1.
 - Similar in use as ABM elements, ABM/I, ABM1/I, ABM2/I and ABM3/I elements have an output which is a current source in the direction shown.
 - Get the Table element from the ABM library. The output of this element on the right side is a voltage (with respect to ground) related to the input voltage by the transfer curve. This transfer curve can be defined point-by-point with increasing input values. Straight-line interpolation is used between two successive points. Make sure to delete all default values for this transfer curve.
 - Other important elements in the ABM library are: filters (BANDPASS, BANDREJ, LOPASS), DIFF, Gain, INTEG, LIMIT, SOFTLIM, MULT and SUM.
- Analog Controller Modeling using Laplace-Domain Transfer Functions
Get a LAPLACE part from the ABM library. Both the numerator and the denominator can be changed by double clicking on them.
 - Specifying Parameters by means of PARAM in Special Library
Get PARAM from the Special library. Double click on it. Enter the name of the parameter and its value in the fields NAME1 and VALUE1 respectively. Up to three parameters can be set up in the PARAM part. In using these

parameters, for example, rather than specifying a numeric value for an inductor, we can specify the parameter name within {} brackets; its value is specified by the PARAM.

- Models for a Power Diode, a MOSFET and an IGBT in EVAL Library

The evaluation version of PSpice has a model for a power diode D1N4002, a MOSFET IRF150, and an IGBT IXGH40N60 in EVAL library. (Models for diode MUR2020, and MOSFET IRF640 are included in PE_LIB1.)

6. Library PE_Lib1: Specially-Designed for Power Electronics (supplied on the Workshop 2003 diskette)

Components in the library of symbols are specially designed for simulation of power electronic circuits and controllers (not a part of the PSpice Install folder). Each of these libraries can contain a maximum of 15 symbols.

6.1 Diodes, MOSFETs, and Switches

D_pwr: This diode has the default values of the built-in PSpice diode model, except the on-state resistance (rs), which can be changed.

MUR2020: This is a high-voltage fast-recovery diode intended for power electronics applications. This model is supplied by manufacturer of the diode.

IRF640: This is a high-voltage MOSFET intended for power electronics applications. The model is supplied by the manufacturer of the MOSFET.

V-SWITCH: A voltage-controlled switch whose on-state resistance (ron) can be changed. Current flow through this switch is bi-directional.

6.2 PowerPole Average Models

CCM_DCM_AVG_2_PORT: An average model of a powerpole valid in both DCM and CCM.

PPW: An alternative formulation of average model of a powerpole. Use in place of CCM_DCM_AVG_2_PORT gives convergence problems.

CCM_AVG_2PORT: An average model of a powerpole valid only in CCM. Simpler model than PPW or CCM_DCM_AVG_2_PORT.

6.3 PWM Controllers

pwm_saw: Produces pulse-width modulated unipolar pulse train by comparing input dc voltage against a sawtooth wave. User supplies dc voltage proportional to duty cycle and the sawtooth (switching) frequency.

pwm_tri: Same function as pwm_saw except a triangular wave is used instead of a sawtooth. Used for inverter circuits.

7. Fourier Analysis

After a time-domain simulation, it is possible to perform a Fourier analysis on one or more of the circuit waveforms. This is accomplished as follows (see Exercise 3): With the schematic on the screen, under the 'Transient' tab of the Simulation setup check Enable Fourier in the "Fourier Analysis" box and provide the necessary information.

Note that the Fourier analysis is performed for the last time period corresponding to the specified center frequency. The phase angle for a sine wave starting (at zero) at

$\text{time} = t_{\text{final}} - t_{\text{period}}$ is zero. The amplitudes are in peak values. After the simulation, the Fourier analysis results are contained in the output file, which can be viewed by selecting 'Examine Output' under the 'Analysis' menu.

8. Parametric Analysis

If an attribute of a component is defined by PARAM as described earlier, then it is possible to perform a parametric sweep. This is accomplished as follows (see Exercise 3): With the schematic on the screen, click on the Simulation Setup icon, select 'Parametric'. Select the global parameter radio button. Give the parameter name and the list of values.

9. Performance Analysis in Probe

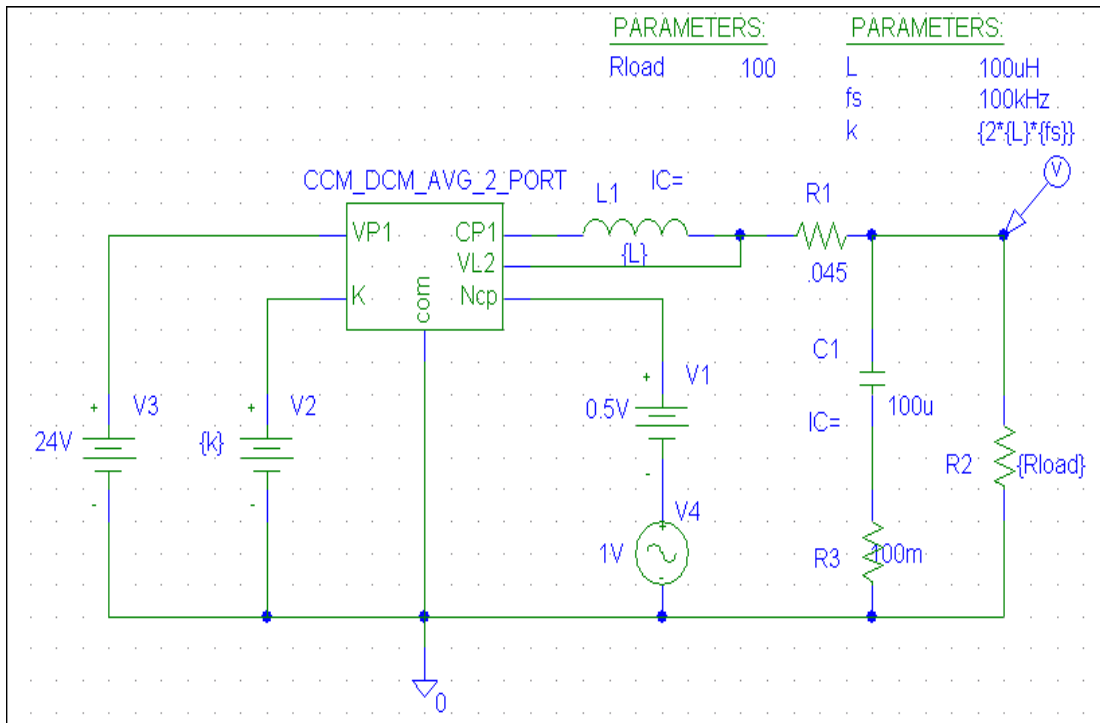
After the results of a parametric analysis are plotted in Probe, pull down the 'Trace' menu and click on 'Performance Analysis...'. Click on 'OK'. Under the 'Trace' menu, click on 'Add'. You will see the built-in Functions or Macros. Choose the desired one and add the trace and other specification, if any, to the Function or Macro. It is possible to use more than one function or Macro in the 'Trace Expression' to evaluate the desired performance. Click on 'OK'.

10. Analysis of Waveforms using built-in Macros within Probe

Any of the traces can be analyzed within Probe by built-in macros such as RMS, AVG, MIN, MAX, etc. For a frequency-domain analysis, macros such as DB and P (for phase) are available.

11. Frequency Domain Analysis

1. In Schematics, open the schematic file Buck_Conv_Freq.sch. The circuit schematic shown below should appear in Schematics window.



2. In the simulation setup, select 'AC Sweep/Noise' as the Analysis type. The sweep type should be decade and the frequency range should be from 10Hz to 1MHz with 101 points per decade.

After running the simulation, the Probe window will show the voltage across the load resistor as a function of frequency. This voltage is the transfer function between the output voltage and the input duty cycle since the ac duty cycle (given by V4) is equal to unity. A more conventional gain and phase plot can also be constructed. First pull down the 'Trace' menu in Probe and select 'Delete All Traces'. Then pull down the 'Trace' menu and select 'Add Trace'. From Functions of Macros, select P(). From the 'Add Traces' select V(R2:2) (the voltage across the load resistor R2). Click on 'OK'. You'll see a Phase plot on your screen. From the 'Plot' menu, select 'Add Plot to

Window'. From the 'Trace' menu, select 'Add Trace' and then DB() from the Functions or Macros. Once again add V(R2:2). Click on 'OK'. You will see a Gain plot.

4. Double click on the X-axis in the bottom plot. Click on 'Axis Variable'. In the 'Trace Expression' window, type in 2π in front of Frequency. Click on 'OK'. Click on 'OK' again.
5. Select cursor icon from the tool bar. Move cursor to 10K. (Note that the x-axis scale is now in rad/s, not in Hz). On the tool bar, click on 'Mark Label' icon.
6. Note: The Parametric Analysis and the Performance Analysis can also be carried out in the frequency-domain.

12. Avoiding and Overcoming Convergence; Other Useful Hints

In modeling of power electronic circuits, convergence and the speed of simulation are two of the biggest problems. The following precautions are suggested.

1. It may be necessary to include R-C snubbers across diodes, thyristors and switches to avoid a sharp discontinuity, which results in PSpice proceeding with a very small time step.
2. Increase the error tolerance ABSTOL to 1u, GMIN to 1u, ITL1 to 400 and ITL4 to 100. It may be necessary to increase RELTOL for 0.01. This can be accomplished as follows: with the schematic on the screen, click on Simulation setup and then on 'Options' tab and change the desired values.
3. Instead of L, use Lr component, which has a resistance in parallel with the inductance. Similarly, use a small resistance in series with large capacitors and voltage sources.
4. Assign small but finite rise and fall times and pulse width to pulsed sources.
5. Connect a large resistance across an element, which seems to be causing a convergence problem.

Floating Nodes: There must be a dc path to ground from each node. Otherwise, an error message will indicate that a particular node is floating. A simple remedy is to connect a large resistance, for example a 1 Meg ohm resistance, from that node to ground, which does effect the circuit performance. At each node, there must be at least two connections, otherwise the run will abort with an error message. A large resistance from that node to ground will correct this problem.

Inductor Loops: If there is a loop involving inductors with zero resistance, an error message will be printed in the .OUT file. Insert a small resistance anywhere in this loop.

Appendix - Descriptions of PSpice Tutorial Circuits

- * **BuckConv_Sw_Model** - Buck converter implemented with a voltage-controlled switch. Intended for transient analysis to show basic converter waveforms.
- * **Buck_MOSFET** - Buck converter implemented with vendor-supplied MOSFET and diode models. Intended for transient analysis to show both basic converter waveforms and details of individual switchings of the MOSFET and diode.
- * **Buck_Conv_Avg** - Buck converter utilizing average model of the powerpole. Intended for transient analysis to show effects of step load change.
- * **Buck_Freq_Response** - Similar to Buck_Conv_Avg except that the circuit also has an ac source to provide a sinusoidal variation to the duty cycle. Intended for swept ac analysis to shown the transfer function versus frequency between the output voltage and the input duty cycle.
- * **Buck_conv_avg_fb_ctrl** - Average model of a buck converter with voltage mode feedback control. Intended for transient analysis to show converter response to a step load change.
- * **Buck-conv-sw-fb_ctrl** - Buck converter implemented with ideal voltage-controlled switch and voltage mode feedback control. Intended for transient analysis to show response to step load change.

- * **DBrdg_Rect** - Full bridge single phase diode rectifier. Intended for transient analysis to show voltage and current waveforms as source inductance is changed and calculation of harmonic components and total harmonic distortion.
- * **Inverter_1PH_BP** - Single phase inverter with bipolar switching. Intended for transient analysis showing sinusoidal output current.