

Instruction Set for Simulating Power Electronics using PSpice™ (Release 9.1) Schematics

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

by
Ned Mohan
Professor of ECE
University of Minnesota
Minneapolis, MN 55455 USA

Reference Book:

First Course on Power Electronics by Ned Mohan (www.MNPERE.com)

Copyright © 2008 by Ned Mohan

Table of Contents

	Page
1. Introduction	3
1.1 Installing PSpice from PE Book CD	
1.2 Simulation as a Three-Step Process	
2. Getting Started Using an Example	3
2.1 Open a Schematic File	
2.2 Simulate	
2.3 Plotting Results using Probe	
3. Getting to know PSpice by an Example	4
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	7
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	9
6. Library First_Course_PED	11
7. Fourier Analysis	12
8. Parametric Analysis	12
9. Performance Analysis in Probe	12
10. Analysis of Waveforms using built-in Macros within Probe	12
11. Frequency Domain Analysis	13
12. Avoiding and Overcoming Convergence; Other Useful Hints	14

1. Introduction

1.1 Installing PSpice

The CD accompanying the Reference Book (First Course on Power Electronics by Ned Mohan and published by www.mnpere.com) contains the files needed for installing the evaluation version of PSpice 9.1. Follow the instructions in the file: Readme_PSpice.doc.

1.2 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 schematics 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:\FirstCourse_PE_Book09\buckconv.sch and click on 'Open' button.

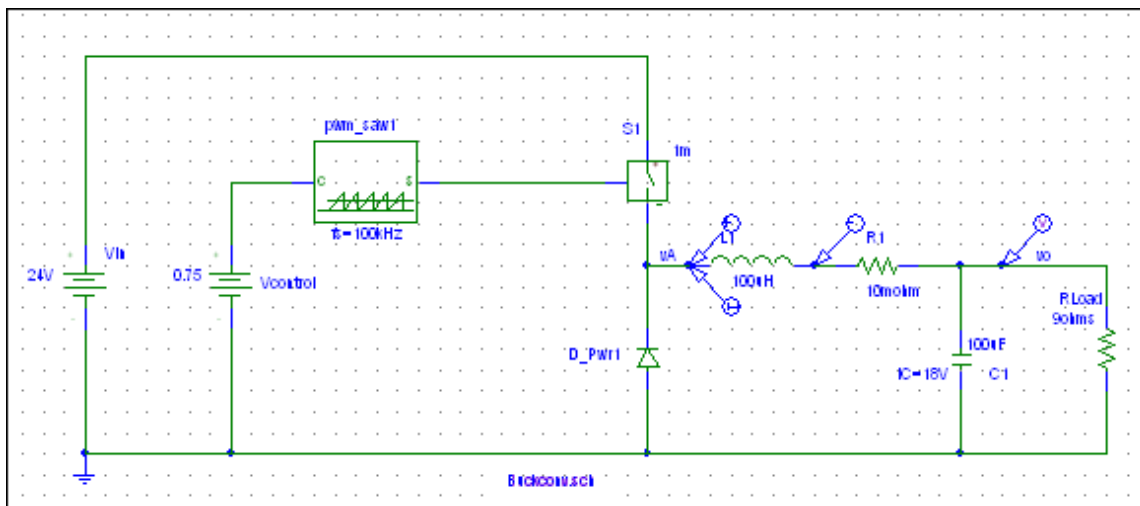



Fig. 1 Schematic file “Buckconv.sch”

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 Buckconv.dat created by the transient simulation.
- Exit probe by clicking on two 'X' buttons on the upper-most right hand corner.

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

- 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 First_Course_PED 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 First_Course_PED. 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'.
- 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 'First_Course_PED.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", 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 First_Course_PED 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. Upto 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 'First_Course_PED': Specially-Designed for Power Electronics

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.

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

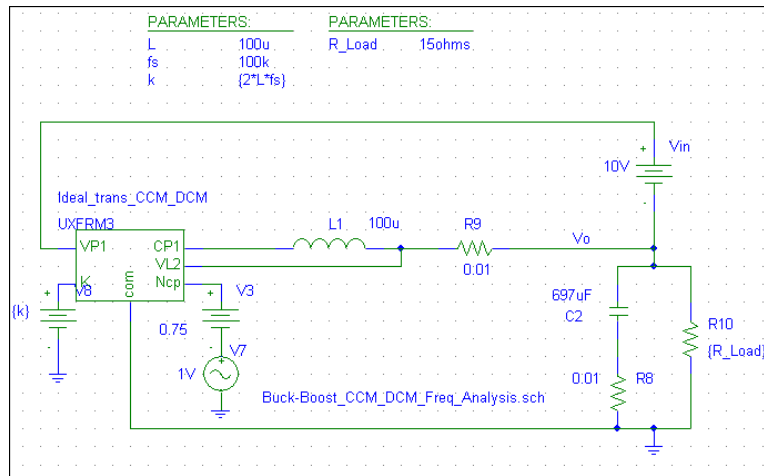
After the results of a parametric analysis are plotted in Probe, pull down the 'Trace' menu and click on 'Performance Analysis...'. Click on 'OK'. Under the 'Trace' menu, click on 'Add'. You will see the built-in Functions or Macros. Choose the desired one and add the trace and other specification, if 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-Boost_CCM_DCM_Freq_Analysis.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.
3. 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 V7) 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(vo). 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(vo). 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 $2*\pi*$ 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.