A Quick Setup and User Guide to Simulating Power Electronics using PSpice™ (Release 9)

1. Installing PSpice

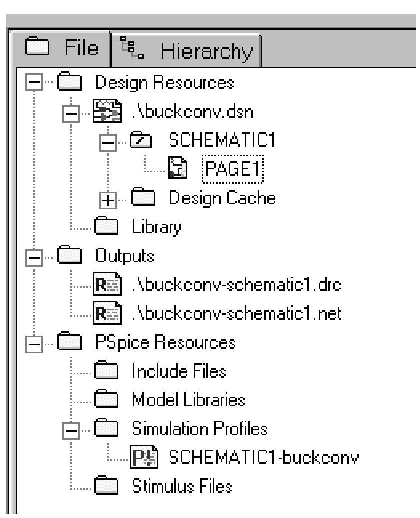
- Insert the CD and install Capture CIS and PSpice, choosing the default option in each step. Complete installation by clicking on "Finish".

2. Copying Power Electronics Files from Accompanying Diskettes

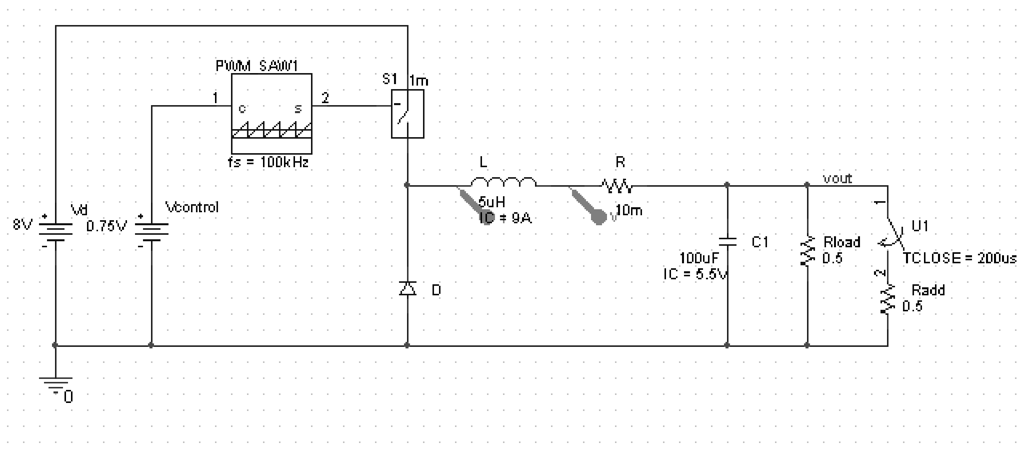
- Copy all the files in the Folder named PE_PSpice_Files on this CD-ROM to a folder on the same drive (for example in a folder called c:\my_PSpice_Files).

3. Open a Project File

- To launch Capture, click on the following: Start - Programs – OrCAD – Capture CIS.
- Under the 'File' menu, click on 'Open' - 'Project…'.
- Navigate or type in the following filename with the full path name, for example, c:\my_PSpice_Files*Buckconv.opj* and click on 'Open' button. This is the highest level project file where all the information about the related files to this example project are stored. Follow down the hierarchy by expanding Design Resources, .\buckconv.dsn, SCHEMATIC1, PAGE1.



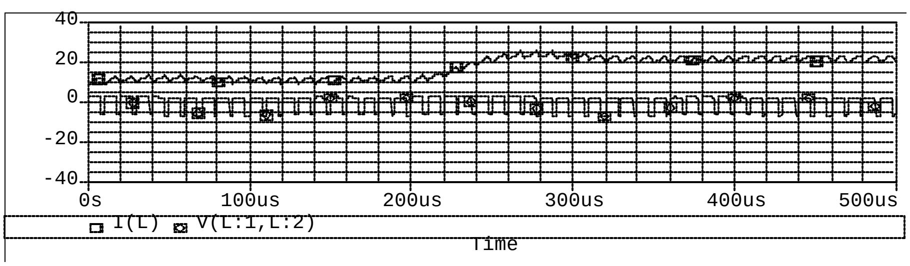
Double click on PAGE1. This should bring the schematic of Example 7 on the screen. For plotting the results, if the markers are not connected, click on the



Current Marker icon on the Tool Bar and place it on the inductor pin. Similarly, click on the Voltage Differential Markers icon and place them across the inductor.

4. **Simulate**

- Click on the run icon to simulate (or under the 'PSpice' menu, click on 'Run'). This simulation is set up to run the plotting program automatically to plot the voltage and current selected by the markers in the schematic at the end of the simulation.



5. **Plotting and Analysis of Plotted Variables using built-in Functions**

Use of Markers is one of the ways to select the variables for display after the program execution. Another way, when the plotting window opens up, is to pull down the 'Trace' menu and click on 'Add Trace...'. You will see a list of Simulation Output Variables and the list of Functions. Any of the variables can be analyzed by the built-in Functions such as RMS, AVG, MIN, MAX, etc. For a frequency-domain analysis, Functions such as DB and P (for phase) are available.

6. **Parametric Analysis**

If an attribute of a component is defined by PARAM as in Example 1, then it is possible to perform a parametric sweep. This is accomplished as follows: With the schematic on the screen, under the 'Analysis' tab of the Simulation Settings, select ‘Parametric

Sweep' under the options menu. Select the global parameter radio button. Give the parameter name and the list of values.

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 Example 1): With the schematic on the screen, under the 'Analysis' tab of the simulation settings, select 'Output File Options'. Check on 'Perform Fourier Analysis' box and provide the necessary information.

Note that the Fourier Analysis is performed for the last period of the specified center frequency. The phase angle for a sine wave starting (at zero) at time=t_{final}-t_{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 'View Output File' under the 'PSpice' menu.

8. **Printing Plotted Waveforms and the Schematic**

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'.

- <u>Taking the waveforms to a word processor</u>: 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.

- <u>Printing the Schematic</u>: 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.