About PSpice-based Examples

The intent of the material presented here on this CD is to provide a quick and hopefully sufficient information for most users to begin to utilize PSpice for analysis and simulation purposes in power electronics courses.

This folder contains the following:

1. A file labeled Guide.PDF. This contains a quick installation and user guide for simulating power electronics using PSpice.
2. A file labeled PSpice_Examples.PDF which contains 27 examples of simulations and associated problems that can be solved by executing the simulation files contained in the sub-folder labeled PE_PSpice_Files.
3. A sub-folder labeled PE_PSpice_Files which contains the simulation files which can be executed using PSpice.

These examples and simulations are designed primarily for educational purposes. They are provided "as is" without any guarantee or warrantee either expressed or implied and no author or distributor of this material will be held responsible for any damage that may possibly result from the use or misuse of this material. The user acknowledges and accepts sole responsibility for his/her usage of this material.

In the Table of Contents, each example refers to the corresponding simulation file within brackets. Each example also refers to relevant section and page numbers in this textbook.

These simulations are designed and tested on Release 9 of the student/evaluation version of PSpice which is a registered trademark of the OrCad Corporation. This evaluation version can be obtained free of charge by requesting it from **www.pspice.com**.