Introduction to PSpiceÂ® Manual, Electric Circuits: Using ORCadÂ® Release 9.1; 2000; James William Nilsson, Susan A. Riedel; 9780130165633; 132 pages; Prentice Hall, 2000

The version of PSPICE available in the Labs is ORCAD version 10.0. The following instructions are also appropriate for version 9.1 which can be downloaded on the ELEC3400 course webpage. Please note that in the version 10.0 there is no â€˜Schematicsâ€™ program. This is an alternative program to draw and simulate circuits with PSPICE. Please use the â€˜Captureâ€™ program in version 9.1 to draw your circuits. This program is compatible with the one in version 10.0. For a more detailed explanation of other features in PSPICE go to the â€˜Helpâ€™ menu in the â€˜Captureâ€™ program and select the â€˜Orcad PSpiceâ€™ opt 1st Tutorial: Introduction to PSpice. q PSpice File Types q Basic Rules q References q Node Designations q Large & Small Numbers q Independent Voltage Sources q Independent Current Sources q Resistors q DC Sweeps & the .PRINT Command. 2nd Tutorial: Simple Dependent Sources. q Voltage Controlled Dependent Voltage Source q Voltage Controlled Dependent Current Source q Current Controlled Dependent Current Source q Current Controlled Dependent Voltage Source q Using PSpice to find the ThÃ©venin Equivalent Circuit. 3rd Tutorial: Subcircuits. q Coding a Subcircuit q Invoking a Subcircuit q Sc UM2167 User manual. OrCAD PSpice model usage instructions. Introduction. This document describes how to use STâ€™s PSpice models available for SMPS devices. The models are useable in the OrCAD system environment of Cadence Design Systems and will not work in other simulation platforms. Furthermore, we recommend using the latest version of OrCAD to avoid convergence problems and speed up the simulation. PSpice models describe the characteristics of typical devices and don't guarantee the absolute representation of product specifications and operating characteristics; the datasheet is the onl