

# Using PSpice to simulate the SPICE examples in Microelectronic Circuits (5<sup>th</sup> Edition) CD Release 2 (January 2004)

This document is a brief guide for using PSpice to simulate the SPICE examples presented in Microelectronic Circuits (5<sup>th</sup> Edition) and available on this CD. The reader is assumed to have a basic working knowledge of SPICE and the PSpice simulator.

Update to CD Release 1:      Corrected the values of the model parameters for parts NMOS0P5 and PMOS0P5 in the sedra\_lib.lib file to match the values given in Table 4.8 (p.335) and used in the SPICE Examples of the textbook.

## 1. Software Packages

The SPICE examples of Microelectronic Circuits (5<sup>th</sup> Edition) were designed in the commercial version of Cadence PSD 14.2 using Capture CIS for schematic entry, PSpice A/D for circuit simulation, and Probe for graphical display and numerical analysis (see Section 1.8 in the textbook). A student version of Capture CIS, PSpice A/D, and Probe is available on this CD. This corresponds to OrCAD Family Release 9.2 Lite Edition from Cadence. Note that, in the (free) student version of PSpice, circuit simulation is restricted to circuits with no more than 10 transistors.

All SPICE Examples of Microelectronic Circuits (5<sup>th</sup> Edition) can be simulated using the student version of PSpice, except Examples 7.6 and 11.5 due to the restrictions on the number of transistors per circuit schematic.

## 2. Getting Started

The SPICE examples of Microelectronic Circuits (5<sup>th</sup> Edition) should be extracted from the */install\_SPICE\_Examples.exe* file on the CD, into a desired location on a local drive. This procedure is automated by clicking on the *Install SPICE Examples* icon from the CD's main menu.

## 3. Running Simulations

The *SPICE\_Examples* folder contains one sub-folder for each chapter in Microelectronic Circuits (5<sup>th</sup> Edition), except Chapter 1. All SPICE examples in a given chapter are grouped into a single Capture project file, ending with a *.opj* extension.

### To open the Capture schematics of a given SPICE Example:

1. Start *Capture CIS* (or *Capture CIS Lite*) from the *Program Menu*
2. Select *File>Open>Project* and chose a *.opj* file in the desired chapter sub-folder

This will open the *Project Manager* window, which provides links to all the SPICE examples in the selected chapter. Click on the + box under *Design Resources* to expand the design contents of the chapter. Each item listed under the *chapter X.dsn* heading is a separate *schematic*, which is simulated separately. Notice that separate *pages* under a given *schematic* are treated as a single netlist and are therefore simulated together.

Each *schematic* can have any number of *simulation profiles* associated with it. A *simulation profile* specifies the type of simulation analysis to be performed on the associated *schematic* (AC, DC, transient, etc.). The simulation parameters (sweep variables, time steps, etc.) are also stored within the *simulation profile*. The *simulation profiles* are shown in the *Project Manager* under *PSpice Resources > Simulation Profiles*

**To run the PSpice simulation of a given Capture schematic:**

1. Right-click on the desired *schematic* and select *Make Root*
2. Right-click on a *simulation profile* for that *schematic* and select *Make Active*
3. Click on *PSpice > Run*

This will automatically invoke the PSpice simulator and the output waveforms will be displayed in Probe.

**To graphically select (before or after the simulation) the output variables to be displayed in Probe:**

1. Expand the desired *schematic* by clicking on the + box in the *Project Manager* window
2. Double-click on the *page* of interest (most *schematics* have only one *page*)
3. Select *PSpice > Markers > Voltage Level* and place the *probe* on the desired node (you can also use the *marker* buttons on the task bar)

## **4. PSpice Libraries and Parts**

The description of the parts used in the SPICE examples (other than the primitive parts such as the passive components, connectors, etc.) is included in 2 library files within the *SPICE Examples* folder on the CD:

1. *sedra\_lib.olb* : contains the symbols for the parts
2. *sedra\_lib.lib* : contains the SPICE netlist associated with each symbol

These library files should not be moved because all SPICE examples have a relative link to these files. Note that you can view the SPICE netlist of a given part by right-clicking on it and selecting *Edit Pspice Model*.