14:332:223 Principles of Electrical Engineering I

Instructions for using PSPICE Tools

Sharanya Chandrasekar

September 23, 2004

1. Getting Started

PSPICE is available on the ECE Computer labs in EE 103 and on all the computing facilities on campus (ARC, LSM, etc.). If you prefer to work from home, you can download a free student version of PSPICE from the website below: www.orcad.com/downloads/demo/default.asp.

2. Creating Schematics

There are two ways to creating SPICE schematics
   (a) Create manual netlists in a text file using simple modeling statements provided by SPICE. Type up the netlist and save the file as <filename.cir>. For further details on the syntax, visit http://dave.uta.edu/dillon/pspice01.htm. If you need further help with this method, contact sharanya@eden.rutgers.edu
   (b) Use the Orcad Capture schematics tool to use a graphical interface drag and drop mechanism to build the schematic and have the tool figure out the netlist from that. This document will cover instructions to use this tool.
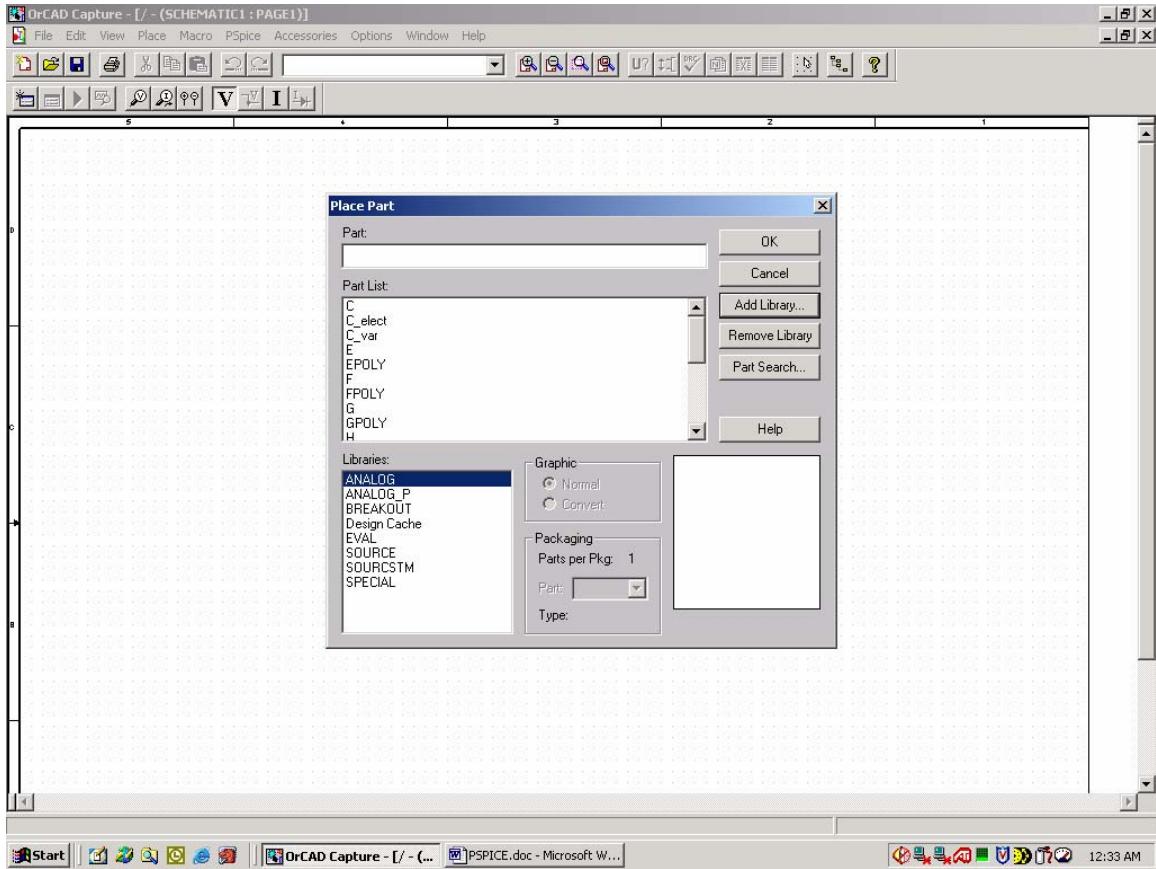
2.1 Drawing Circuits
Circuits can be created in the Orcad Capture Tool.

2.1.1 Creating a New Project
   1) Start up the program Orcad Capture.
   2) Click on File -> New -> Project
   3) On the dialog box, provide a project name, location and select check box for Analog Mixed Signals. Click OK
   4) On the next dialog box, select create blank project.
2.1.2 Placing Parts
   1) Click on Menu Place → Part

2) If nothing is listed under libraries, then click on Add Library button. On the resulting dialog box, select all the files under ../../Capture/library/Pspice. Most of the parts that would be used in this lab can be found in the Analog and Source libraries. Typing its name can search for a particular part. Also, the most commonly used symbol for it is displayed in the preview pane.

3) Select a part from the list and click on OK. Multiple instances of that part can now be placed on the schematic. Right click on the part to rotate mirror etc to organize the circuit elements.

4) To change the circuit variable applicable to a part (resistance, voltage etc), double click on the part and type in the value.

5) Click on Place → Wire to inter-connect the devices and complete the circuit. Be sure to connect all points together till a junction is formed. If required, zoom in to make sure if a contact has been made.

6) After building the circuit, place GND (ground) part in the circuit. This is usually the negative node of a power source. Click Place → Ground.

7) If in the resulting dialog box, there are no libraries listed, click on the add libraries button. Add the source and sourcetm libraries under ../../capture/library/pspice directory.
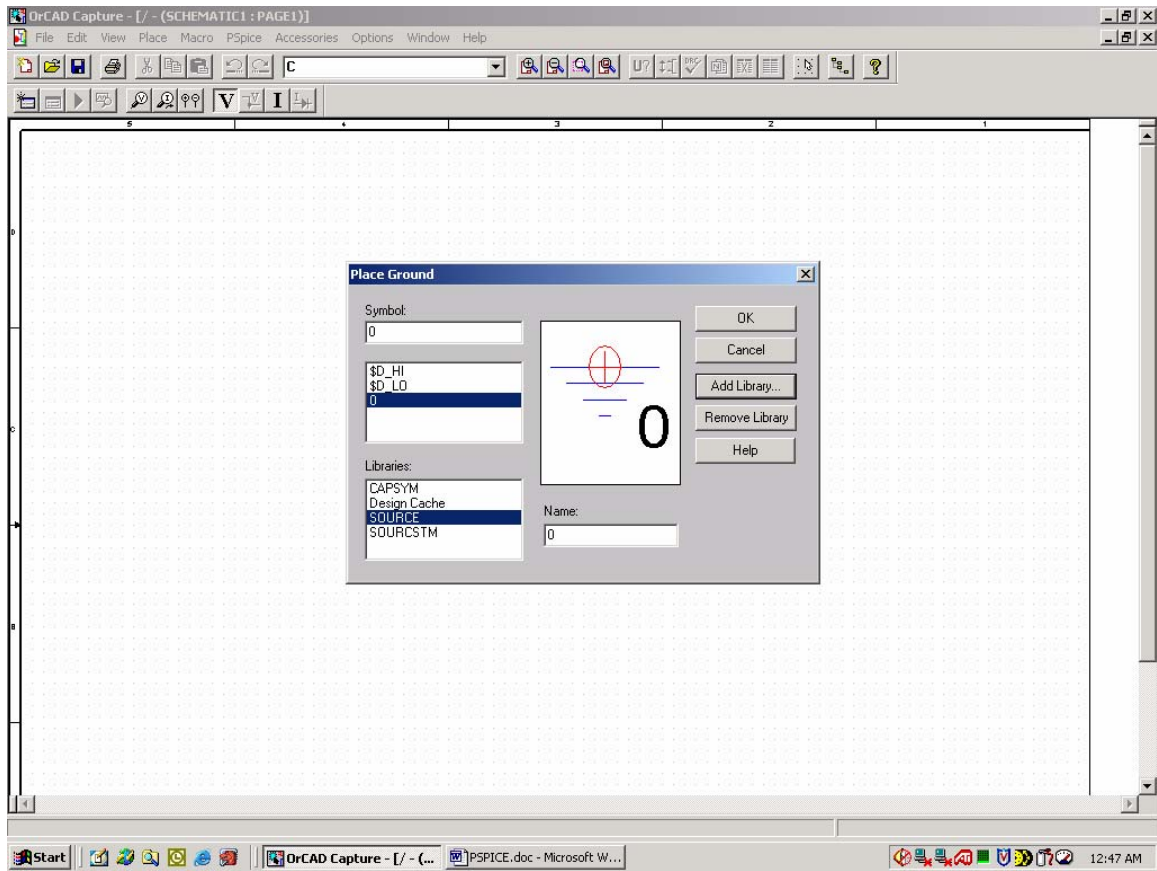
8) Select the part 0 under source library for GND
9) To check if the schematic is correct, click on Pspice → Create Netlist. Any errors will be reported at this time. Go back to the schematic and correct it. The most common error encountered is 'floating node'. This usually means that there is some problem in interconnects.
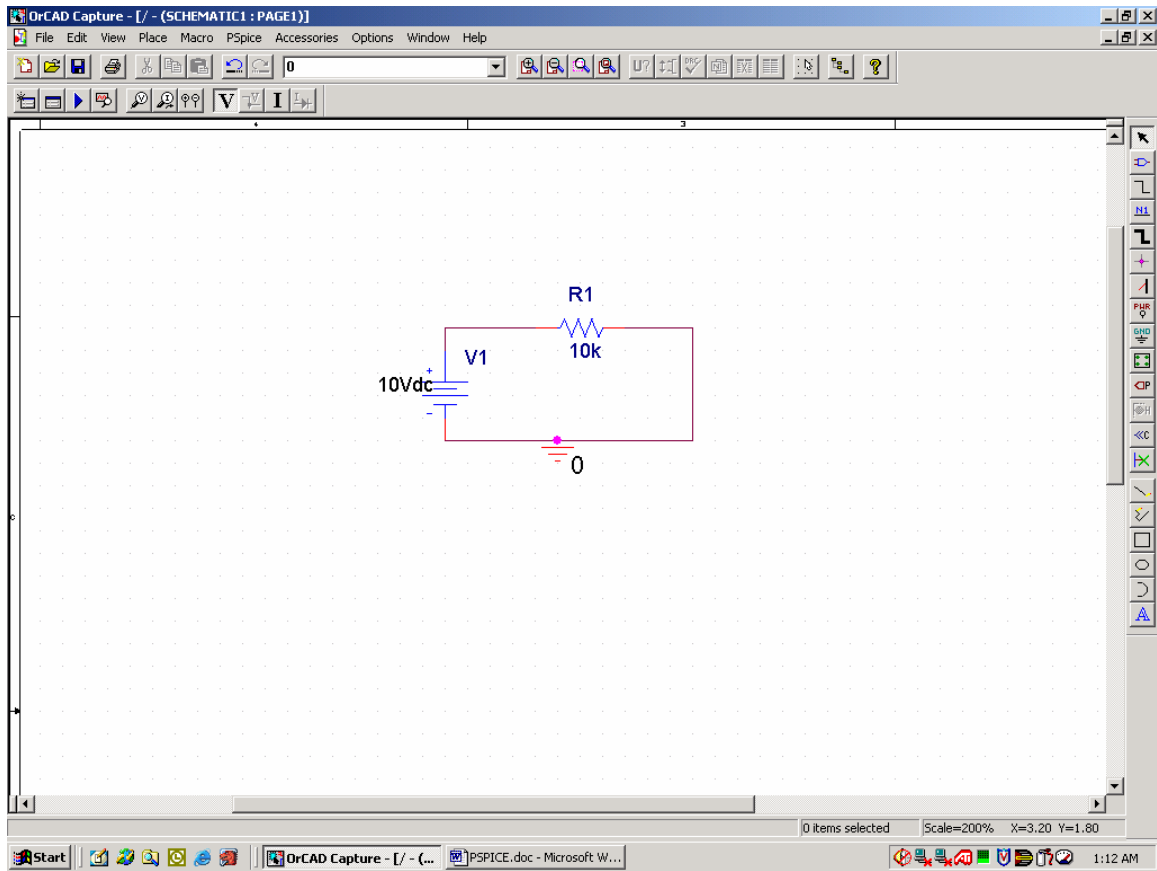
2.2 Simulating the circuit

1) After correcting all netlist errors start by creating a simulation.
2) Click Pspice → New Simulation Profile. Provide name for the simulation and select none in the inherit from list. Click on Create.
3) Select Analysis type. Below is a brief description of each type of analysis.
   a. Bias Point Analysis – Compute DC voltages and currents in DC circuits
   b. DC Sweep – Observe (Plot) the behavior of a circuit variable when an input source is changed.
   c. Transient analysis – Observe (Plot) the behavior of a certain circuit variable with time
   d. AC Analysis – Compute frequency response etc in AC circuits.
4) Enter simulation information according to the analysis type chosen. Click OK.
5) To run the simulation, select Pspice → Run. A new simulation window opens and runs the simulation. Any errors are reported here. Correct schematic or the simulation settings and re-run simulation.
6) To view the output file, click Pspice → View Output File.
7) To view simulation plots (DC sweep, Transient and AC analysis), on the simulation window, click Trace → Add Trace. In the resulting dialog, select the variables to be plotted and click OK.