

Capture and PSpice Reference Guide

The following is a step by step guide for beginning work with Capture and PSpice. It details the basic functions and abilities of the two programs, as well as provides a reference for correctly setting up simulations, component libraries, component values, and measuring probes, within the scope of the Phys. 261 course.

Setting up a schematic

- Open Capture: ALL PROGRAMS->CADENCE 16 ->DESIGN ENTRY CIS
- Respond to “Allegro PCB Librarian XL” with OK
- When CIS window opens, maximize the screen. You can readjust this later in order to view both schematics and analysis results simultaneously.
- Create new project: FILE->NEW->PROJECT
- Type in a name, choose “analog or mixed A/D”
- Set location by browsing to a folder in your MY DOCUMENTS folder. This location is where all of the relevant files associated with this project will be stored. Since there are multiple files per project, it’s a good idea to create a new folder for each project in whichever directory you choose.
- To “create PSpice project”, choose “blank project”
- When Schematic window opens, scale it to upper 2/3 of CIS window.
- To add parts (resistors, capacitors, voltage sources, etc.) to your schematic, click on “Part Placement” icon
- If this is the first time Capture has been run on the computer, you will need to manually add the part libraries to the interface. For the types of circuits encountered in Phys 261, the parts will be located in the “analog” and “source” libraries. To add these, click “Add library” and choose “pspice/analog” and “pspice/source”
- To add an active circuit component, such as a DC voltage source (representing a battery), select it from the “source” library and click OK.
- Place the source by positioning it on the schematic and clicking, then press “esc” to stop placing parts.
- Double clicking on the part in the schematic will open up the part properties window, which you should move to the 1/3 area left under your schematic. This allows you to quickly examine the various values of components while still viewing your circuit layout.
- To add a passive circuit component, such as a resistor, select it from the “analog” library and place it in the same manner as other components.

- To rotate components before placement, press the “R” key on the keyboard.
- Each component will have two labels that appear nearby. These represent the component name with respect to your schematic (e.g. R1 for resistor 1) and the current value of that component (e.g. 3k next to a resistor is $3k\Omega$, while 5Vdc is 5 volts for a DC source). In general, units (Ω , V, A) are not required to be specified. However, magnitudes must be given using the case sensitive notation:
 - k –kilo
 - m –milli
 - M –mega
 - u –micro (instead of μ)
 - n –nano
 - p –pico
- To connect components together, click on “wire” icon and then click once at each location to draw in a connecting wire. Wires will not connect to each other if you draw across them—you must click once at the junction to form a node.
- Every circuit must be given a 0V ground reference, or PSpice will not be able to analyze them. To add a ground, click on the “ground” icon
- Add “source” library and choose and place “0 V Ground”

Setting up and Running Pspice

- Choose: PSPICE->NEW SIMULATION PROFILE
- Name the profile (generally the same name as your project file) and choose inherit “none”
- PSpice can perform 4 types of analyses: Time domain, AC sweep, DC sweep, and Bias. For most projects, this is the only Simulation setting that you will need to modify. Each analysis type requires you to input several parameters that define the simulation. As an example, the following steps detail how to set up a DC sweep.
- Choose “DC sweep” from the drop down menu
- Select whether you’re sweeping across values of a voltage or current source
- Type in the part name of that particular source (e.g. V1)
- Enter a start, end, and increment value for your sweep. For example, if you wanted to do a DC sweep for a voltage source from 0-5V taking data every 0.5V, then you would enter 0 start, 5 end, 0.5 increment. Note that numbers in the Simulation window must be entered in full decimal form (3000 instead of 3k, or 0.006 instead of 6m)
- Click OK
- Place measurement probes as needed on circuit. Current probe must be at a node. (You will see that + or – current will depend on which node is probed.)

- Click “RUN” icon.
- The PSpice analysis window should open automatically. It will display the relevant data for each probe that was placed, generally in the form of a plot. Your schematic will still be open in the background. You can place new probes after the simulation has ended to see new data added to the plots. However, PSpice has calculated ALL the data inherent to the circuit, so you can manually add plots for whichever voltage or current you want:
 - Choose PLOT->ADD PLOT
 - Click on the parameter you want plotted from the window.
 - The parameter will appear in the bottom line of the window.
 - Click OK
 - Note that the text line at the bottom also allows for various mathematical formulas to be entered to act on your parameters, so you can easily plot values like V_1/V_2 .

Outputting Data

- Often, you will want to compare PSpice data with data that you have collected by hand, usually with a spreadsheet program like Excel. You can export point by point data from PSpice to a text file, which can then be imported into another program.
- Choose FILE->EXPORT->TEXT
- Set “. . .” to send file to an easily found folder of your choice (the same directory as your project would be a wise choice)
- (After first save, this selection is remembered)
- Run Microsoft Excel and choose FILE->OPEN and open saved data file
- Click “finish” to finalize import of data file.