SPICE QUICK-START GUIDE

(PSpice 9 & 10 – Student Version)

A step-by-step procedure for starting OrCAD PSPICE for the first time and drawing a circuit.

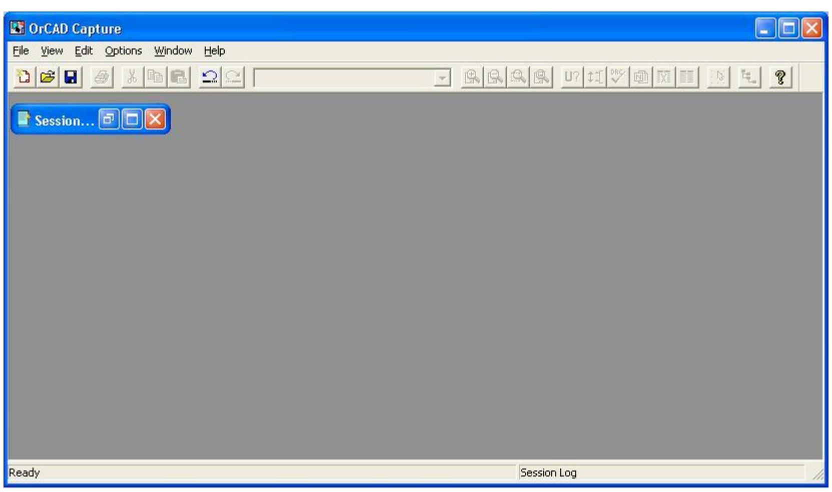
DOWNLOAD & INSTALLING

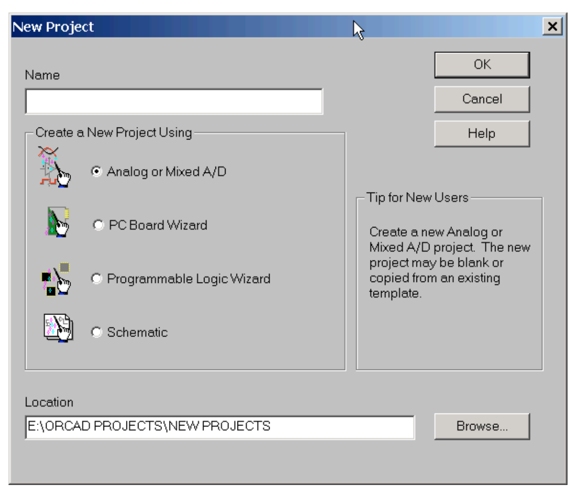
After downloading the ZIP file (http://www.orcad.com/Product/Simulation/PSpice/eval.asp or search the OrCAD site for an updated file; class WebCT page may also have a download), open it and move down inside it to the file Setup.exe. Double click on this file to start the installation and accept all the default values. This will install a suite of 4 tools: PSpice A/D, Capture, Schematics, and PSpice Optimizer. These programs provide tools for drawing circuit diagrams and simulating the voltages and currents at points under various test conditions.

STARTING SPICE & INITIALIZING

Once the software has been installed correctly, start it through the Windows **start** menu -> **All Programs** -> **PSpice Student** -> **Capture Student**. The names **PSpice Student** and **Capture Student** may be different depending on which version of PSPICE is installed. You are looking for an executable program with 'Capture' in its name.

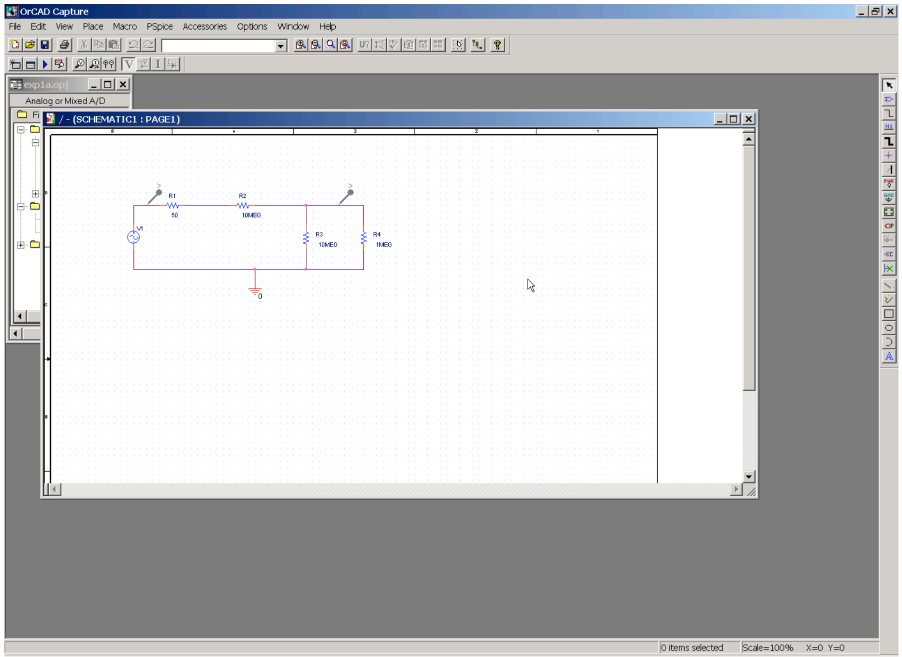
After starting up **Capture**, a window labeled *OrCAD Capture* (shown below - left) will open with menus at the top. Select **File** -> **New** -> **Project…** to create a new schematic (shown below – right). Enter a name for your new project in the *Name* field and select **Analog or Mixed A/D** under the label *Create a New Project Using*, optionally changing the location where the project will be stored by clicking **Browse…** at the bottom of the window under the *Location* label and then clicking **OK**. A new window will open labeled *Create PSpice Project* requiring you to make a selection basing your new project on an existing one or starting a new one. Select **Create a blank project**.





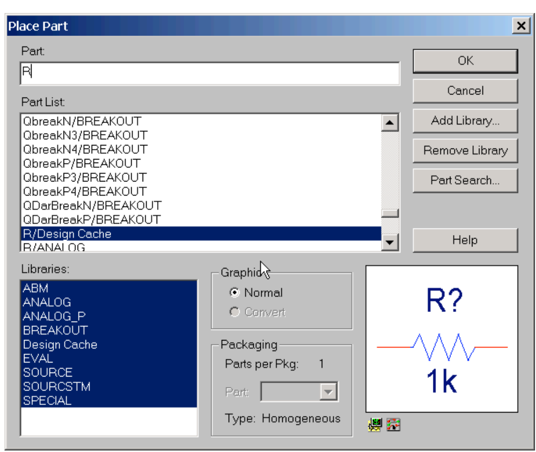
NOTE: Wherever your new project is stored on you hard disk, a number of files will be created with various file extension types. The main file will have the name you provided in the *Name* field and an extension .opj. Double clicking on this file will start **Capture** and open up this project.

Now a new schematic window will open labeled *(SCHEMATIC1 : PAGE1)* (as below – but with a blank schematic) in which you will be able to draw your circuit for simulation. To do this you must select parts from the library and place them on the sheet, interconnect them with wire, select points for which you want the simulator to graph voltages or currents, and run the simulation after specifying required parameters.



It is suggested that the automatically drawn label box in the lower right hand corner of the screen be deleted to allow the page to be scaled better on the screen and easier to manipulate. Do this by dragging the cursor across the box to highlight it and then hitting the <Backspace> key on the keyboard.

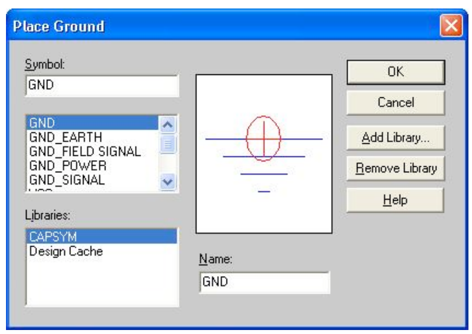
Before you can place parts on the schematic for the first time you will need to add libraries to the parts list. After this time the libraries should remain available to all other projects and circuits. To add a part, select **Place -> Part…**, type **<Shift>P**, or click on the small **AND gate** that is the second buttons down in the column on the right side of the *OrCAD Capture* window. This opens up a *Place Part* window. To add the parts libraries click on **Add Library…**, select all the files in the *Browse File* window, and click **Open**. The *Browse File* window will close and return you to the *Place Part* window. Selecting a library under *Libraries:* at the bottom will display all the available devices of that library in the *Part List:* field at the top. Selecting a part will display its schematic symbol in the lower right rectangle. Selecting all the libraries at once (dragging the cursor from the top item to the bottom) will list the parts in all the libraries together (see below).



CREATING A SCHEMATIC

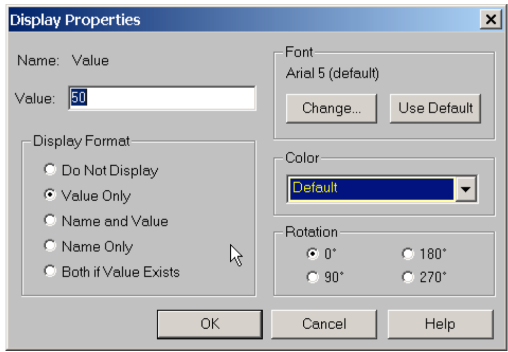
Most of the actions used in creating schematics are accomplished by using the command buttons at the top and right side of the *OrCAD Capture* window. Moving the mouse arrow until it is located over any button will display its function in text.

Every circuit requires a ground reference point so this should be one of the first parts on every drawing. Unfortunately this can cause some confusion and prevent a simulation from working if not done correctly. Also, there are several different ground symbols that may not be in any of the parts libraries. To get a ground symbol, use **Place** -> **Ground…**, **<Shift>G**, or click on the **GND symbol** that is the ninth button down in the column on the right side of the *OrCAD Capture* window. In the *Place Ground* window that opens select **CAPSYM** under *Libraries:* and click on **GND**, the top item in the list. Its schematic symbol will show up in the center rectangle (see below). Click **OK** to close the window and move the cursor over the sheet to the position where you want to place the ground. Click once to attach the symbol. If you need several grounds you may move to the other positions and click wherever you want a ground. When you are done, right drag on the drawing down to **End Mode** in the menu that pops up and release. This completes the use of the part and allows you to place another or begin to connect wires. Any symbol may be rotated by first clicking on it to select it and typing **R** or then right dragging down to **Mirror Horizontally**, **Mirror Vertically**, or **Rotate** in the pop-up menu. The part rotates 90° CCW each time **Rotate** is selected. <Control>Z will undo the last action, as usual.

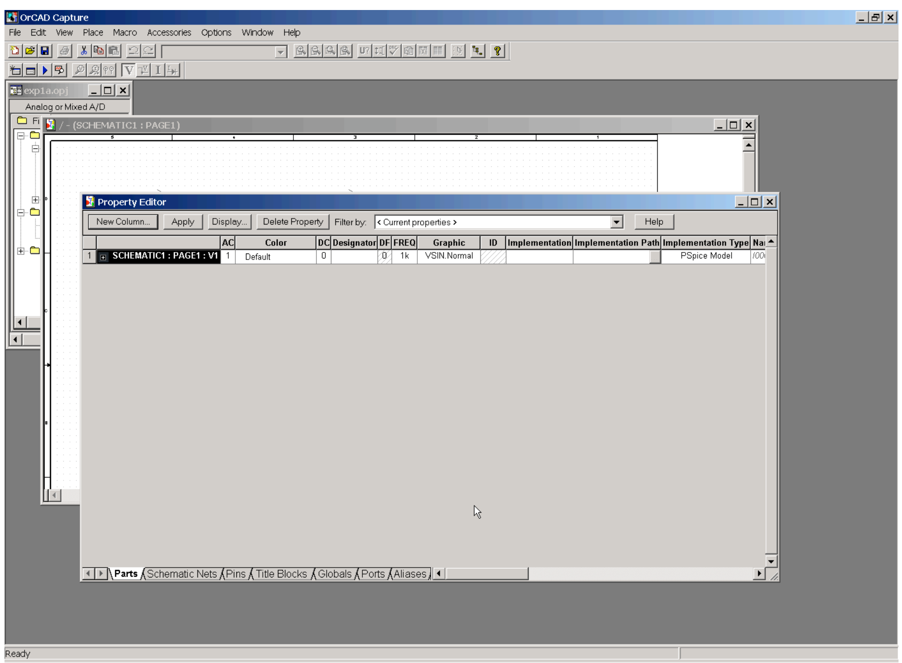


Before a circuit can be simulated, the ground(s) needs to be referenced by defining its voltage. This is an unusual 'feature' since most grounds are assumed to be 0 V automatically! Do this by double clicking on the ground to open up the *Property Editor* window. Make sure the *Globals* tab at the bottom is selected and in the 2-row table displayed, change the entry under the column labeled *Name* from GND to 0 (the digit 0). If this is not done, attempts to run the simulator on the circuits will result in errors saying there is no data to plot. This step is not necessary in version 10 of PSpice.

Place resistors, capacitors, and inductors in the circuit where you like and use **Rotate** or the **R** key to change their orientation. These components are in the **ANALOG** library with names **R**, **C**, and **L** or in the **ANALOG_P** library as **r**, **c**, and **l**. Click on the part in the *Part List:* or type its name in the *Part:* field. Click **OK** to get back to the drawing sheet where you can move the pointer to the desired location and left click to place the part there. Typing **<Esc>** or clicking on the **Place part** button will also end the mode. Each component is given a name in the order its type was placed on the diagram to insure uniqueness. Special attention must be paid to the orientation of capacitors and inductors when nonzero initial conditions are used. A capacitor drawn in its default orientation with a positive initial voltage will have the negative side to the left labeled 1 and positive side to the right labeled 2. A single rotation of 90° will place the negative lead on the bottom and the positive lead on the top. There is no mark on the capacitor indicating this polarity and you will need to keep track of it on your own by the 1 – 2 labels. Similarly, an inductor with a positive initial current will have it defined moving from the left lead (1) to the right lead (2) in the inductor's default orientation. Initial conditions are set in capacitors and inductors by double clicking on them to open up the *Property Editor* window. Select the *Parts* tab at the bottom of the window, in the grayed area under the column labeled IC type in the desired numerical value, click on the IC column heading, click on the **Display…** button, select **Name and Value** under *Display Format* in the pop-up *Display Properties* window, select **OK** in the *Display Properties* window to close it, and click on the '**X**' in the upper right corner to close the *Property Editor* window. A new line will appear next to the component in the drawing displaying its initial condition value. It is probably a good idea to automatically assign initial conditions to all capacitors and inductors in a circuit. The components placed on the drawing are assigned default vales [1k (Ω) for resistors, 1n (F) for capacitors, and 10u (H) for inductors]. These values are changed by double clicking on the value field of the part displaying the default value and entering the desired value in the *Value:* field in the pop-up *Display Properties* window (see below).



After placing the devices on the drawing, sources may be added to provide driving waveforms to the circuit or battery power to active devices. Several different voltage and current sources are available in the **SOURCE** library. Most sources have parameters which must be specified using their *Property Editor* window that pops up when the source is double clicked. Most of the values should be self-explanatory (see below).

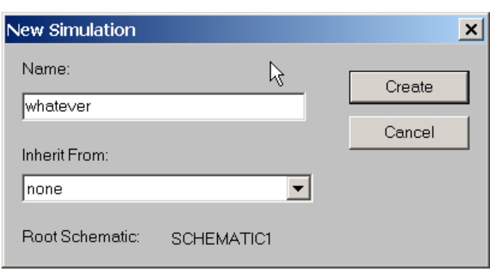


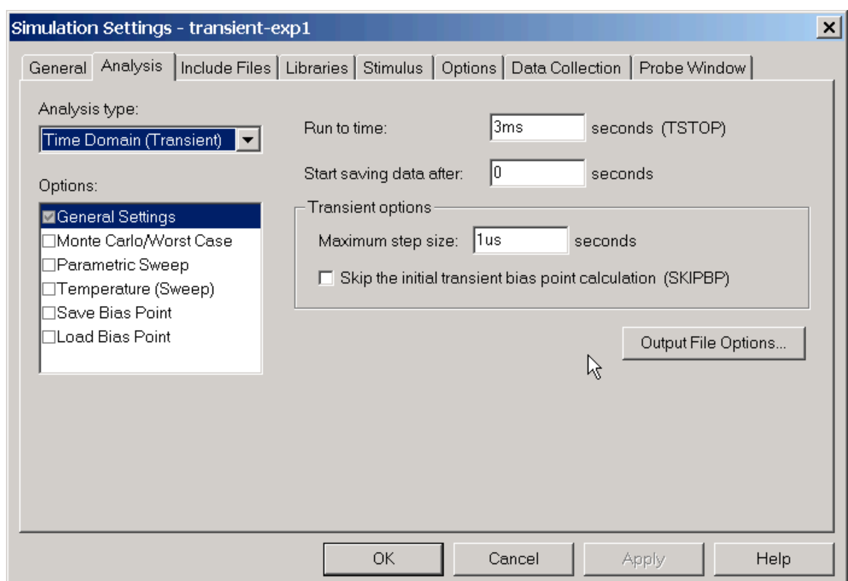
Once all the parts have been placed on the drawing, connect them together using the wire tool, the third button down in the column on the right side of the window. With the cursor changed to a cross-hair, click on the first point to be connected and move the mouse to the second point. Use the little wire on each device as its connection point. If you need to turn a corner, click where the wire should bend 90°. Move the cursor to the final destination and click once more on the red dot that should appear indicating a connection then double click to end the wire. More wires can be added following the same procedure. Note that whenever you connect more than one wire at some point, a dot will appear there indicating a connection. It is possible for wires to cross one another without connecting if you choose, in which case no dot will appear. A wire can be deleted by selecting it and hitting <Backspace> or <Delete>. You may need to deselect the wire tool and use the **Select** tool, the top button down in the column on the right side of the window. Parts may also be moved around by clicking and dragging them to a new location. All wires should remain connected. This is a good procedure to find unconnected parts.

RUNNING THE SIMULATOR

Before simulations can be done, desired values to be plotted must be selected, such as voltages at nodes or currents through devices. Differential voltages across parts or between any two points on the circuit may also be selected. Voltage and current probes can be selected from the toolbar at the top of the screen designated with a 'V' or 'I'. The text names are **Voltage/Level Marker** and **Current Marker**. Select the desired probe, place it on the wire whose voltage or current you would like plotted, and click. If you miss the wire you will see an error message and will need to try again. Probes may be rotated, but only through the keyboard **R** key. Right click and select **End Mode** or hit **<Esc>** to quit.

Previous to the first time a simulation is run, a simulation profile must be created. Select **PSpice -> New Simulation Profile** (or click on the button at the top of the screen), enter a name in the *Name:* field in the *New Simulation* window (see below – left) and click on **Create**. A new *Simulation Settings* window will open with the *Analysis* tab selected (see below – right). There are 3 main types of simulations that can be selected under *Analysis type*: **Time Domain (Transient)**, **DC Sweep**, and **AC Sweep/Noise**. For each simulation type, a series of curves may be plotted if it is desired to change a circuit parameter, such as resistance or capacitance, over a range of values to observe the progression of the family of waveforms.



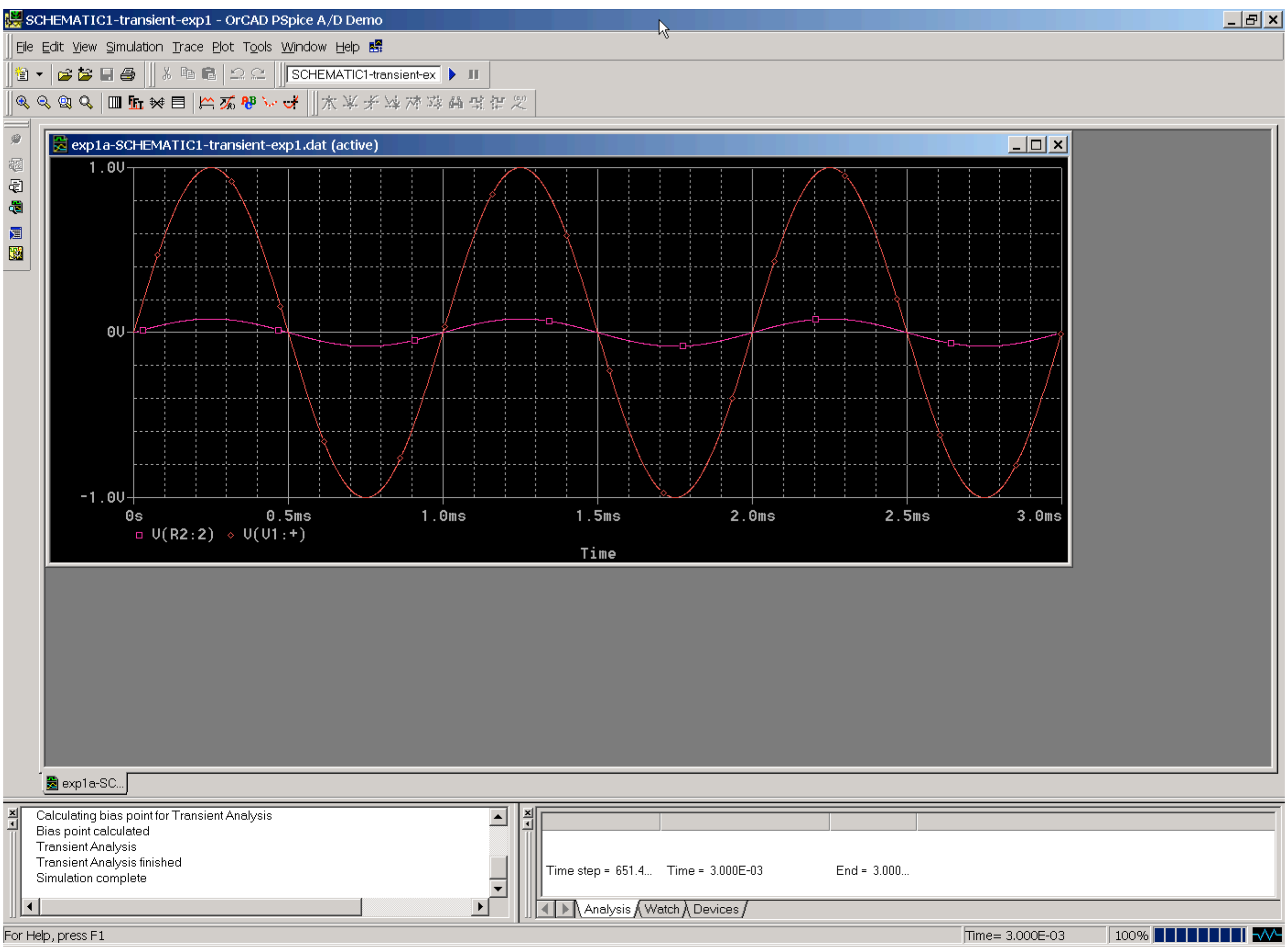


Time Domain (Transient) is used most frequently to evaluate circuit waveforms as functions of time. This is described more fully below.

DC Sweep offers a few options for DC analysis. A voltage or current source may be swept over a range of values to show how other voltages and currents in the circuit change with the supply's DC value using the **Primary Sweep** under *Options:* and selecting either **Voltage source** or **Current source** under *Sweep variables*. The source's name must be entered in the in the name field. Alternatively, by selecting **Global parameter**, a parameter such as a resistor's value may be varied instead of a source value. The value needs to have been specified as a variable (e.g. as {Radj}) in the component's value field and also included under a PARAMETER list (see details in the USING PARAMETER LISTS section below). A range may be specified with start, end, and increment values, or a list of specific values may be entered under *List:* for sources or parameters. A slightly different method for parameter sweeping is used in combination with **Time Domain** and **AC Sweep** analysis, which will be discussed later.

AC Sweep/Noise is used to generate Bode plots of a circuit's performance, graphing magnitude and phase responses as a function of s (frequency). More details follow in the sections USING PARAMETER LISTS & SWEPT VARIABLES and PLOTTING FREQUENCY RESPONSE WITH AC SWEEP.

When selecting **Time Domain (Transient)** analysis, the most important parameter here is TSTOP, the stop time of the simulation. The default value of 1000ns should be changed to whatever is appropriate, probably around several milliseconds. Most simulations will be done using Time Domain (Transient), the type of time waveform that would be obtained on an oscilloscope. The *Maximum Step Size* is set to speed up analysis. PSpice will keep cutting its step size down until it achieves some standard for accuracy. For transient analysis, this will sometimes cause it to take a very long time to finish. The analysis will begin saving data at 0 seconds. These parameters should usually be left at their default values. Click on the **OK** button when you are finished putting in the numbers. After creating a profile, use **PSpice** -> **Edit Simulation Profile** to change any settings. Now a simulation can be started by selecting **PSpice** -> **Run** or clicking on the blue triangle icon in the toolbar at the top of the screen, which has now become active along with the rest of the buttons in that group. If everything is working correctly, a new application window will open with the plotted waveforms of the selected voltages or currents vs. time (see next page). If not you will get a nearly undecipherable listing of errors that may or may not be helpful in tracking down the problem(s) in the circuit. See the TROUBLESHOOTING section at the end of this guide.



It is possible to set up many kinds of analyses using the *Simulation Settings* window. You will only use these features later in the semester, so you may choose to skip to the next section for now. Since you have already told PSpice for which nodes you want to know voltages, it will produce a plot with the signal you have asked for. You should now go back and add another voltage level arrow at the location that represents the output of any sources used. Be sure you have the correct location. A very nice feature that **Capture** gives us, that was not available in the past, is that the voltage plots and the voltage probe markers will be the same color, so it will be easy to determine which is which. There are other types of analysis that can be done. To determine the frequency response of a circuit, we can use AC analysis. Click on the **Edit Simulation Settings** button. Set the *Analysis type:* to **AC Sweep/Noise** and set values for the start and end frequencies.

The *AC Sweep Type* is typically chosen as **Logarithmic** - **Decade** since frequency effects usually only become obvious when we change orders of magnitude. This generates a log scale for frequency. The start frequencies and end frequencies are chosen to cover an interesting range. Usually this range is selected from some knowledge of the expected performance of the circuit. However, since we are assuming that we know very little about this circuit, we can set the range to be roughly that covered by the HP function generator. Note that 15MHz had to be written as 15MEG, since PSpice uses both lower case m and upper case M to mean milli. We have to make one more change before we can do AC analysis. Double click on any AC voltage sources again to bring up their attribute spreadsheet. You will now have to give a value to AC. This can really be anything, but make it equal to the sine wave amplitude 100mV. In general, it is not a bad idea to do this from the beginning to avoid problems when the analysis type is changed.

USING PARAMETER LISTS & SWEPT VARIABLES

Normally, component values are defined right next to the part in the displayed field below the part type & count label. There are two other methods commonly used to define values that provide a little more flexibility during the design process where the values of one or more parts may be adjusted or varied throughout a range while the circuit's waveforms are observed. The first method is to use a parametric list with the **PARAM** part in the **SPECIAL** library and creating an entry in the table where the values are defined. To implement this method, the part whose value is to be varied is entered as a global parameter enclosed in brackets. For example, a resistor's value might be changed to {Radj} from the default value of 1k. Now the global parameter Radj must be entered into the parameter table by double clicking on the PARAMETERS: part in the drawing and clicking on **New Column** in the *Property Editor* window. In the *Add New Column* window that pops up enter the global variable name in the *Name:* field, e.g. Radj, and a default value in the *Value:* field, e.g. 10k, and click on **OK**. To get the entry to show up in a list on the schematic, select the column just created in the in the *Property Editor* window (Radj), click **Display…**, then select **Name and Value** in the *Display Properties* window that pops up and click on **OK**. Close the *Property Editor* window by clicking on the '**X**' in the upper right hand corner. A new entry in the parameter table should now be seen. The circuit may be simulated and the values of the global variable defined in the parameter list will be used. If a global variable is left undefined, an error message will appear when a simulation is attempted.

The second method to define a part's value may be used in place of or in addition to the PARAMETERS: method. In this method the global variable is defined in the *Simulation Settings* window opened by **PSpice -> Edit Simulation Profile**. Selecting the *Analysis* tab and checking **Parametric Sweep** under *Options:* displays some new fields. Select **Global parameter** under *Sweep variable* and enter the name of the global parameter, in the *Parameter name:* field, e.g. Radj to continue the example. After clicking on **Value list** under *Sweep type*, the list of values to be applied to the global parameter is entered into the field next to the **Value list** option, separated by commas, e.g. 1k, 5k, 50k. Click on **OK** to close the window. Now when the simulation is run, three sets of waveforms will be plotted on the same time axis, one corresponding to each resistor value specified here. At any time the sweep variable feature may be turned off in the *Simulation Settings* window, but you must insure that that the global parameter is defined elsewhere, such as in a parameters list.

Attaching the proper units to the variables used in part values, parameter lists, and parametric sweeps in simulations is optional. In general, any numeric value may be augmented by a letter modifier such a k, m, p, etc. No space should appear between the number and the letter and the letters are case insensitive. The accepted SPICE modifiers and their values are listed below:

k (or K)	10^3	
meg (or MEG)	10^6	NOTE: M is used for 10^{-3}
g (or G)	10^9	
m	10^{-3}	
u	10^{-6}	
n	10^{-9}	
p	10^{-12}	
f	10^{-15}	

Units of Henrys are designated by H and Farads by F. Resistance is assumed to be in Ohms and no letter designator is used. For the most part, units are ignored on numerical values for resistors, capacitors, and inductors. They exist only for the user's benefit since the simulation program will always assume the correct units for a given component.

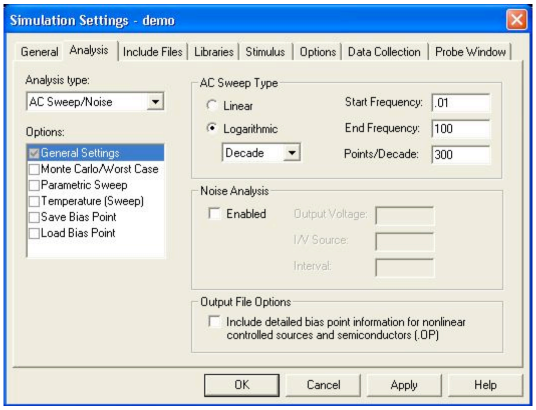
PLOTTING FREQUENCY RESPONSE WITH AC SWEEP

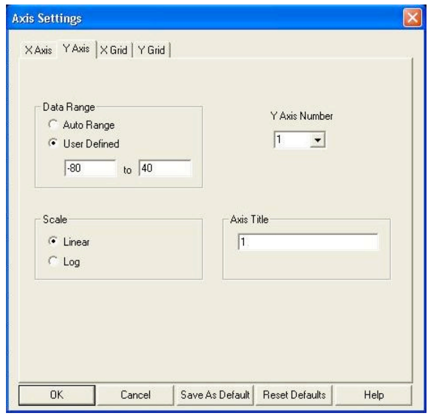
In addition to plotting the values of voltages and currents as functions of time, PSPICE allows you to display the AC response of a circuit vs. frequency. In order for this to work, the circuit must be set up correctly. This means that the swept source is either an AC voltage (VAC) or current (IAC) from the SOURCE library and the probes are dB Magnitude or Phase, voltage or current probes. Instead of selecting the normal voltage or current probes, click on the main *Capture* menu: **PSpice -> Markers -> Advanced -> *Probe***, where ***Probe*** is **dB Magnitude of Voltage**, **dB Magnitude of Current**, **Phase of Voltage**, or **Phase of Current**. Other options available permit the displaying of voltages and currents in rectangular coordinates (real and imaginary values) instead of polar coordinates (magnitude and phase).

Under the simulation settings (**PSpice** -> **Edit Simulation Profile**), change the *Analysis type:* to **A C Sweep/Noise** (see below - left). If you haven't created a simulation profile, go back to the section RUNNING THE SIMULATIOR. With *Options:* set to **General Settings**, pick either **Linear** or **Logarithmic** under *AC Sweep Type* and specify the *Start Frequency:*, *End Frequency:*, and the number of data points to be plotted per decade (*Points/Decade:*). Logarithmic scales do not allow a start frequency of 0, and the linear scale seems to have a problem with it too, so use a small value of 0.1 or 0.01 instead. For these examples make sure *Noise Analysis* is **not** enabled. Click OK to save your values and close the window.

After placing dB magnitude and phase probes on the circuit and inserting the AC voltage and/or current sources, the simulator may be run as before by clicking on the blue triangle or selecting **PSpice** -> **Run**. Unfortunately, the simulator defaults to plotting magnitude and phase on the same axis, so a large phase change of –180° may cause a small magnitude change of 60 dB to be compressed. It is possible to fix this in the simulator's graphing routine but may take a little adjusting to get it to look the way you may like. Other signals may be added to the plot if the simulation output window is left open while new magnitude or phase probes are added to the schematic.

The PSpice simulator's output plotting function is extremely flexible and allows the user to customize plots. Many options exist for labeling axes, scaling axes, adding grid marks, and adding color. A few basic options will be discussed here but you are encouraged to explore some of the other features on your own. To separate magnitude and phase traces onto their own axes, click on **Plot -> Add Plot to Window**. Click on the name below the bottom axis of the value you want to move to the new axis, for example *VDB* (its color should change when it is selected), and type **<Ctrl>C** to copy it, click on the top axis (make sure the side label '**SEL>>**' is pointing to the top axis), then type **<Ctrl>V** to paste it. That value may then be removed from the bottom axis by selecting the variable name again and typing **<Delete>**. If you are not happy with the auto-scaling on either axis, change it by clicking on the plot to change (to get **SEL>>** to point to it) and then double clicking on the **SEL>>** to open the *Axis Settings* window (see below - right). Select the desired tab (**X Axis** or **Y Axis**) and click on **User Defined** under *Data Range* and enter the lower and upper values to be used. Click on **OK** to close the window. The *Axis Settings* window is used to turn on grid lines, major and minor axis tick marks and other options for marking the plots.





PARTS LISTS AND COMMONLY USED PARTS

Selecting a part for placement displays a list of provided libraries of parts. Listed below are the most common libraries and the types of components they contain.

ABM: Functional blocks whose outputs are transcendental functions of inputs, common constant values, summing junctions, controlled sources, etc.
ANALOG: Fixed and variable resistors and capacitors, inductors, transformers, controlled sources, and relays.
ANALOG_P: Short list of most commonly used resistors, capacitors, and inductors.
BREAKOUT: Semiconductor devices (diodes, zener diodes, BJT transistors, MOS transistors, DAC and ADC blocks.
Design Cache: List of devices used in the current design.
EVAL: List of standard commercial parts such as logic gates, op amps, and discrete components as well as switches, edge connectors, and jumper points.
SOURCE: Digital clocks, AC & DC voltage and current sources, batteries, sine wave generators, and repeated pulse waveforms,
SOURCESTM: Short list of clock generators, voltage and current sources.
SPECIAL: I/O functions and parameter lists.

The most commonly used components used in class work are:

TYPE	NAME	LIBRARY	
Ground	GND	CAPSYM	(use **Place** -> **Ground…**, or **<Shift>G**)
Battery	VDC	SOURCE	
AC voltage	VAC	SOURCE	

Voltage source	VSRC	SOURCE	
Voltage sine wave	VSIN	SOURCE	
Current source	ISRC	SOURCE	
Current sine wave	ISIN	SOURCE	
Pulse train	VPULSE	SOURCE	(V1 is V off, V2 is V on, TD is time delay before start, TR & TF are rise and fall times)
Fixed resistor	R	ANALOG	
Variable resistor	R_var	ANALOG	(Change the 'set' field of *Parts* tab from 0.5 to 1 in properties – double click part; far right of list)
Fixed capacitor	C	ANALOG	
Variable capacitor	C_var	ANALOG	
Electrolytic capacitor	C_elect	ANALOG	
Inductor	L	ANALOG	
VCVS	E	ANALOG	
CCCS	F	ANALOG	
VCCS	G	ANALOG	
CCVS	H	ANALOG	
Transformer	XFRM_LINEAR	ANALOG	
741 op amp	uA741	EVAL	
Switch (closes at time T)	Sw_tClose	EVAL	
Switch (opens at time T)	Sw_tOpen	EVAL	
Transistor 2N2222	Q2N2222	EVAL	
Diode	D	EVAL	
Parametric list	PARAM	SPECIAL	

TROUBLESHOOTING

Every schematic entry GUI has technical problems and OrCAD PSPICE is no exception. Many times a correctly entered circuit will fail to produce simulation results through no fault of the user. These lead to extremely frustrating situations, but a little patience and a knack for troubleshooting will allow the user to triumph. First, it is important to make sure all grounds have been renamed '0'. Next make sure all wires and components are actually connected to the points they are supposed to be by clicking on them and moving them around. If the wire or component is properly connected, moving it around will stretch the wire to where it is attached but not break the connection. In some cases merely moving inconsequential objects around on the drawing, such as labels or parameter lists, will allow a simulation to start working. In general, it is always a good idea to build and test a small sub-circuit rather than build the entire circuit and then test it. For example, a circuit containing an op amp may be built with just the op amp first and tested by applying a known signal source (VSIN) to it and verifying its output. Only after this is working should more components be added to the circuit. The simulator may get confused if many parts are added and deleted to a schematic and wires are rerouted. Occasionally the circuit will become corrupted and it will be very difficult to determine where the problem is. At this point it may be faster to delete the entire circuit and start over rather than try to find the error.

ON-LINE REFERENCES

http://www.ecs.umass.edu/ece/ece211/ECE211_PSpice_Tutorial2.pdf
http://www.hull.ac.uk/engineering/teaching/57027/OrCAD.pdf